R&S®ZNB
Vector Network Analyzers
User Manual



1173916302
Version 73

ROHDE&SCHWARZ
Make ideas real

This document applies to the following vector network analyzer models and their options:

- R&S® ZNB4, 9 kHz to 4.5 GHz, 2 test ports, order no. 1311.6010K22
- R&S® ZNB4, 9 kHz to 4.5 GHz, 2 test ports, order no. 1334.3330K22
- R&S® ZNB4, 9 kHz to 4.5 GHz, 4 test ports, order no. 1311.6010K24
- R&S® ZNB4, 9 kHz to 4.5 GHz, 4 test ports, order no. 1334.3330K24
- R&S® ZNB8, 9 kHz to 8.5 GHz, 2 test ports, order no. 1311.6010K42
- R&S® ZNB8, 9 kHz to 8.5 GHz, 2 test ports, order no. 1334.3330K42
- R&S® ZNB8, 9 kHz to 8.5 GHz, 4 test ports, order no. 1311.6010K44
- R&S® ZNB8, 9 kHz to 8.5 GHz, 4 test ports, order no. 1334.3330K44
- R&S® ZNB20, 100 kHz to 20 GHz, 2 test ports, order no. 1311.6010K62
- R&S® ZNB20, 100 kHz to 20 GHz, 2 test ports, order no. 1334.3330K62
- R&S® ZNB20, 100 kHz to 20 GHz, 4 test ports, order no. 1311.6010K64
- R&S® ZNB20, 100 kHz to 20 GHz, 4 test ports, order no. 1334.3330K64
- R&S® ZNB26, 100 kHz to 26.5 GHz, 2 test ports, order no. 1334.3330K63
- R&S® ZNB26, 100 kHz to 26.5 GHz, 4 test ports, order no. 1334.3330K65
- R&S® ZNB40, 10 MHz to 40 GHz, 2 test ports, order no. 1311.6010K72
- R&S® ZNB40, 100 kHz to 40 GHz, 2 test ports, order no. 1311.6010K82
- R&S® ZNB40, 100 kHz to 40 GHz, 4 test ports, order no. 1311.6010K84
- R&S® ZNB43, 100 kHz to 43.5 GHz, 2 test ports 2.92 mm, order no. 1334.3330K92
- R&S® ZNB43, 100 kHz to 43.5 GHz, 2 test ports 2.4 mm, order no. 1334.3330K93
- R&S® ZNB43, 100 kHz to 43.5 GHz, 4 test ports 2.92 mm, order no. 1334.3330K94
- R&S® ZNB43, 100 kHz to 43.5 GHz, 4 test ports 2.4 mm, order no. 1334.3330K95

© 2026 Rohde & Schwarz
Muehldorfstr. 15, 81671 Muenchen, Germany
Phone: +49 89 41 29 - 0
Email: info@rohde-schwarz.com
Internet: www.rohde-schwarz.com
Subject to change – data without tolerance limits is not binding.
R&S® is a registered trademark of Rohde & Schwarz GmbH & Co. KG.
The terms HDMI, HDMI High-Definition Multimedia Interface, HDMI trade dress and the HDMI Logos are trademarks or registered trademarks of HDMI Licensing Administrator, Inc.
Intel®, the Intel logo, and Intel® Core™ are trademarks of Intel Corporation or its subsidiaries.
MIPI® marks and logos are service marks owned by MIPI Alliance, Inc. and any use of such marks by Rohde & Schwarz is under license.
All other trademarks are the properties of their respective owners.

1173.9163.02 | Version 73 | R&S®ZNB

Throughout this manual, R&S® is abbreviated as R&S.

Contents

1 Safety and regulatory information....15

1.1 Safety instructions....15

1.2 Warning messages in the documentation....18

1.3 Korea certification class A....18

2 Welcome to the R&S ZNB....19

2.1 Documentation overview....19

2.1.1 Getting started manual....19

2.1.2 User manual and help....19

2.1.3 Service manual....19

2.1.4 Instrument security procedures....20

2.1.5 Printed safety instructions....20

2.1.6 Specifications document and brochure....20

2.1.7 Release notes and open source acknowledgment (OSA)....20

2.1.8 Application notes, application cards, white papers, etc....20

3 Getting started....21

3.1 Preparing for use....21

3.1.1 Lifting and carrying....21

3.1.2 Unpacking and checking....21

3.1.3 Choosing the operating site....21

3.1.4 Setting up the product....22

3.1.5 Considerations for test setup....24

3.1.6 Connecting the analyzer to the AC supply....25

3.1.7 Switching the instrument on and off....26

3.1.8 Standby and ready state....26

3.1.9 Windows operating system....27

3.1.10 Minimizing the VNA application....28

3.1.11 Connecting external accessories....29

3.1.12 Remote operation in a LAN....31

3.2 Instrument tour....35

3.2.1 Front panel....36

3.2.2 Rear panel....41

3.3 Operating the instrument 44
3.3.1 Manual operation 45
3.3.2 Control elements of the application window 48
3.3.3 Touchscreen gestures 54
3.3.4 Working with dialogs 56
3.3.5 Handling diagrams, traces, and markers 57
3.3.6 Entering data 61
3.3.7 Scaling diagrams 65
3.4 Performing measurements 70
3.4.1 Transmission S-parameter measurement 71
3.4.2 Reflection S-parameter measurement 77

4 Concepts and features 79
4.1 Basic concepts 79
4.1.1 Global (persistent) settings 79
4.1.2 Recall sets 80
4.1.3 Traces, channels and diagrams 80
4.1.4 Sweep control 82
4.1.5 Data flow 89
4.2 Screen elements 93
4.2.1 Display elements of a diagram 93
4.2.2 Dialogs 104
4.2.3 Trace formats 108
4.3 Measurement results 116
4.3.1 S-parameters 117
4.3.2 Reference impedances 119
4.3.3 Impedance parameters 121
4.3.4 Admittance parameters 125
4.3.5 Wave quantities and ratios 126
4.3.6 Unbalance-balance conversion 129
4.3.7 Stability factors 134
4.3.8 Group delay 134
4.4 Operations on traces 135
4.4.1 Limit check 135

4.4.2 Trace files....143
4.4.3 Memory-mapped trace data transfer....152
4.5 Calibration....154
4.5.1 Calibration types....155
4.5.2 Calibration standards and calibration kits....168
4.5.3 Calibration pool....174
4.5.4 Calibration state labels....175
4.5.5 Automatic calibration....175
4.5.6 Scalar power calibration....187
4.5.7 SMARTerCal....193
4.5.8 Recommended calibration methods....196
4.5.9 Parallel calibration of multiple channels....196
4.5.10 Joining calibrations....197
4.6 Offset parameters and de-/embedding....197
4.6.1 Offset parameters....197
4.6.2 Embedding and deembedding....203
4.7 Optional extensions and accessories....216
4.7.1 Time domain analysis....218
4.7.2 Distance-to-fault measurements....231
4.7.3 Frequency conversion measurements....233
4.7.4 Receiver bandwidth 10 MHz....243
4.7.5 Frequency resolution 1 mHz....244
4.7.6 Noise figure measurement....244
4.7.7 Measurement uncertainty analysis....248
4.7.8 SNP assistant....251
4.7.9 Eazy de-embedding based on IEEE 370....253
4.7.10 In-situ de-embedding....255
4.7.11 Smart fixture de-embedding....255
4.7.12 Delta-L 4.0 PCB characterization....256
4.7.13 Health and utilization monitoring service (HUMS)....257
4.7.14 Bias tees....258
4.7.15 Second internal generator....258
4.7.16 Precision frequency reference....259

4.7.17 GPIB interface....259
4.7.18 Device Control....259
4.7.19 Handler I/O (universal interface)....260
4.7.20 RFFE GPIO interface....260
4.7.21 Additional removable system drive....262
4.7.22 Extended power range....262
4.7.23 Extended dynamic range....263
4.7.24 Receiver step attenuators....264
4.7.25 R&S ZNB40 reflector module with ESD limiters....264
4.7.26 DC inputs....264
4.7.27 USB-to-IEC/IEEE adapter....267
4.7.28 External power meters....267
4.7.29 External generators....269
4.7.30 External switch matrices....271
4.7.31 External DLLs....282
4.7.32 R&S ZNXSIM....284

5 GUI reference....289

5.1 Function keys and softtools....289

5.2 Meas softtool....291

5.2.1 S-Parameters....292
5.2.2 Mixer Measurements....322
5.2.3 Intermodulation....333
5.2.4 Noise Figure....348
5.2.5 Time Domain....356
5.2.6 Extended Time Domain....374
5.2.7 Signal Integrity Apps....410
5.2.8 External DLLs....433

5.3 Format softtool....438

5.4 Scale softtool....444

5.4.1 Scale Values tab....444
5.4.2 Scale Coupling tab....447
5.4.3 Zoom tab....447

5.5 Trace Config softtool....449

5.5.1 Traces tab ... 449
5.5.2 Mem tab ... 455
5.5.3 All Mem & All Data tab ... 458
5.5.4 Math tab ... 459
5.5.5 Trace Statistics tab ... 465
5.5.6 Smooth & Shift & Hold tab ... 471
5.5.7 Infinite Averaging tab ... 474
5.5.8 Trace Data tab ... 475
5.6 Lines softtool ... 484
5.6.1 Limit Test tab ... 484
5.6.2 Ripple Test tab ... 493
5.6.3 Circle Test tab ... 497
5.6.4 Display Circle tab ... 501
5.6.5 Horiz. Line tab ... 502
5.7 Marker softtool ... 503
5.7.1 Markers tab ... 503
5.7.2 Marker Props tab ... 506
5.7.3 Marker Search tab ... 509
5.7.4 Multiple Peak tab ... 514
5.7.5 Target Search tab ... 516
5.7.6 Bandfilter tab ... 518
5.7.7 Set by Marker tab ... 522
5.7.8 Info Field tab ... 524
5.7.9 Marker Coupling tab ... 525
5.8 Stimulus softtool ... 526
5.8.1 Stimulus tab ... 526
5.8.2 Power tab ... 528
5.8.3 Time Domain tab ... 529
5.9 Power Bw Avg softtool ... 531
5.9.1 Source Power tab ... 531
5.9.2 Receiver Power tab ... 532
5.9.3 Bandwidth tab ... 533
5.9.4 Average tab ... 534

5.10 Sweep Softtool 536
5.10.1 Sweep Params tab 536
5.10.2 Sweep Type tab 539
5.10.3 Trigger In tab 552
5.10.4 Trigger Out tab 557
5.10.5 Sweep Control tab 558
5.11 Cal softtool 562
5.11.1 Start Cal tab 562
5.11.2 Cal Devices tab 633
5.11.3 Power Cal Settings tab 648
5.11.4 Use Cal tab 655
5.11.5 Cal Utilities tab 660
5.11.6 METAS Cal tab 660
5.12 Channel Config softtool 670
5.12.1 Channels tab 670
5.12.2 Port Config tab 685
5.12.3 Mode tab 698
5.12.4 Pwr Cal Settings tab 703
5.13 Offset Embed softtool 703
5.13.1 Offset Embed dock widget 704
5.13.2 Offset tab 737
5.13.3 One Way Loss tab 743
5.13.4 Single Ended tab 745
5.13.5 Port Sets tab 748
5.13.6 Balanced tab 751
5.13.7 Ground Loop tab 754
5.13.8 Differential Match tab 757
5.13.9 Open de-/embedding file dialog 759
5.13.10 Config tab 760
5.13.11 Deembed Assistant tab 761
5.13.12 Delta-L tab 771
5.14 File softtool 778
5.14.1 Recall sets tab 779

5.14.2 Favorites tab ... 783
5.14.3 Print tab ... 784
5.14.4 Printer Setup dialog ... 785
5.14.5 Trace Data tab ... 787
5.14.6 More tab ... 787
5.14.7 Configure Additional Simulation Data dialog ... 788
5.14.8 Report tab ... 792
5.15 Tools softtool ... 794
5.15.1 External Tools tab ... 794
5.16 Display softtool ... 795
5.16.1 Diagram tab ... 795
5.16.2 Split tab ... 799
5.16.3 Config tab ... 801
5.16.4 View Bar tab ... 808
5.16.5 Touchscreen tab ... 809
5.17 Setup softtool ... 809
5.17.1 Setup tab ... 810
5.17.2 Freq. Ref. tab ... 841
5.17.3 Remote Settings tab ... 842
5.17.4 Power Meter tab ... 844
5.17.5 Generator tab ... 850
5.17.6 Switch Matrix tab ... 854
5.17.7 Inline Cal System tab ... 862
5.17.8 Port Setup tab ... 864
5.18 Help softtool ... 865
5.19 Additional function keys ... 866

6 Remote control ... 868
6.1 Introduction to remote control ... 868
6.1.1 Remote control via USB ... 869
6.1.2 Starting a remote control session ... 870
6.1.3 GPIB Explorer ... 870
6.1.4 Switchover to remote control ... 872
6.1.5 Combining manual and remote control ... 875

6.2 Messages ... 876
6.2.1 Device messages (commands and device responses) ... 876
6.2.2 SCPI command structure and syntax ... 876
6.2.3 SCPI parameters ... 880
6.3 Basic remote control concepts ... 882
6.3.1 Traces, channels, and diagram areas ... 882
6.3.2 Active traces in remote control ... 883
6.3.3 Initiating measurements, speed considerations ... 884
6.3.4 Addressing traces and channels ... 885
6.4 Command processing ... 886
6.4.1 Input unit ... 886
6.4.2 Command recognition ... 887
6.4.3 Data base and instrument hardware ... 887
6.4.4 Status reporting system ... 888
6.4.5 Output unit ... 888
6.4.6 Command sequence and command synchronization ... 888
6.5 Status reporting system ... 890
6.5.1 Overview of status registers ... 891
6.5.2 Structure of a SCPI status register ... 892
6.5.3 Contents of the status registers ... 894
6.5.4 Application of the status reporting system ... 900
6.5.5 Reset values of the status reporting system ... 903
6.6 Automating tasks with remote command scripts ... 904
6.6.1 The SCPI recorder ... 905
6.6.2 How to determine the required SCPI command ... 907
6.6.3 How to create and export SCPI scripts ... 908
6.6.4 How to open the context sensitive help for the SCPI commands ... 910
7 Command reference ... 911
7.1 Special terms and notation ... 911
7.1.1 Upper- vs. lower-case ... 912
7.1.2 Special characters ... 912
7.1.3 Parameters ... 912
7.1.4 Numeric suffixes ... 912

7.2 Common commands....913
7.3 SCPI command reference....917
7.3.1 CALCulate commands....917
7.3.2 CONFigure commands....1175
7.3.3 CONTrol commands....1183
7.3.4 DIAGnostic commands....1208
7.3.5 DISPlay commands....1210
7.3.6 FORMat commands....1241
7.3.7 HCOPy commands....1242
7.3.8 INITiate commands....1247
7.3.9 INSTrument commands....1251
7.3.10 MEMory commands....1252
7.3.11 MMEMory commands....1254
7.3.12 OUTPut commands....1300
7.3.13 PROGram commands....1305
7.3.14 [SENSe:] commands....1308
7.3.15 SOURce commands....1482
7.3.16 STATus commands....1533
7.3.17 SYSTem commands....1536
7.3.18 TRACe commands....1586
7.3.19 TRIGger commands....1589
7.4 HUMS and service date commands....1596
7.4.1 SNMP/REST commands....1596
7.4.2 System information commands....1600
7.4.3 Device tags commands....1610
7.4.4 Utilization commands....1611
7.4.5 Service date commands....1618
7.5 R&S ZVR/ZVABT compatible commands....1620
8 Programming examples....1647
8.1 Basic tasks....1647
8.1.1 Typical stages of a remote control program....1647
8.1.2 Channel, trace and diagram handling....1650
8.2 Condensed programming examples....1656

8.2.1 Path-independent remote control programs....1657
8.2.2 Trace and diagram handling....1657
8.2.3 Using markers....1666
8.2.4 Data handling....1668
8.2.5 Calibration....1673
8.2.6 Mixer measurements....1681
8.2.7 RFFE/GPIO interface programming....1683
8.2.8 Fixture modeling....1685

9 Error messages and troubleshooting....1687
9.1 Errors during firmware operation....1687
9.1.1 Asynchronous errors....1688
9.1.2 Errors during measurement....1688
9.2 Errors during firmware installation/update....1689
9.3 Collecting information for technical support....1691
9.4 Contacting customer support....1692

10 File and data management....1693
10.1 Accessing, retrieving and handling data....1693
10.2 Storage locations....1695
10.2.1 Firmware application files....1695
10.2.2 Firmware-generated operational data....1695
10.2.3 User-generated operational data....1696

11 Transporting....1697

12 Maintenance, storage and disposal....1698
12.1 Replacing fuses....1698
12.2 Recalibration and repair....1698
12.3 Cleaning....1698
12.4 Storage....1698
12.5 Disposal....1698

13 Annexes....1700
13.1 Firmware update....1700
13.1.1 Update information....1700
13.1.2 Firmware downgrade....1700

13.1.3 Firmware installation 1701
13.2 System recovery 1701
13.3 Interfaces and connectors 1701
13.3.1 Rear panel connectors 1702
13.3.2 LAN interface 1703
13.3.3 GPIB interface 1704
13.3.4 Handler I/O (universal interface) 1707
13.3.5 RFFE GPIO interface 1715
13.4 Showroom mode 1718
13.5 ENA emulation commands 1719
Glossary: Frequently used terms 1727
List of commands 1733
Index 1765

1 Safety and regulatory information

The product documentation helps you use the product safely and efficiently. Follow the instructions provided here and in the following sections.

Intended use

The product is intended for the development, production and verification of electronic components and devices in industrial, administrative, and laboratory environments. Use the product only for its designated purpose. Observe the operating conditions and performance limits stated in the specifications document.

Target audience

The target audience of this document includes developers and technicians, administrators and maintenance personnel performing RF signal measurements. The required skills and experience of the users depend on the test setup and application of the product.

Where do I find safety information?

Safety information is part of the product documentation. It warns you of potential dangers and gives instructions on how to prevent personal injury or damage caused by dangerous situations. Safety information is provided as follows:

- In Section 1.1, "Safety instructions", on page 15. The same information is provided in many languages in printed format. The printed "Safety Instructions" for "Mains-Powered Products, Not Heavy" (document number 1171.1771.99) are delivered with the product.
- Throughout the documentation, safety instructions are provided when you need to take care during setup or operation.

1.1 Safety instructions

Products from the Rohde & Schwarz group of companies are manufactured according to the highest technical standards. To use the products safely, follow the instructions provided here and in the product documentation. Keep the product documentation nearby and offer it to other users.

Use the product only for its intended use and within its performance limits. Intended use and limits are described in the product documentation such as the specifications document, manuals and the printed "Safety Instructions" document. If you are unsure about the appropriate use, contact Rohde & Schwarz customer support.

Using the product requires skilled persons or specially trained personnel. These users also need sound knowledge of at least one of the languages in which the user interfaces and the product documentation are available.

Reconfigure or adjust the product only as described in the product documentation or the specifications document. Any other modifications can affect safety and are not permitted.

Never open the casing of the product. Only service personnel authorized by Rohde & Schwarz are allowed to repair the product. If any part of the product is damaged or broken, stop using the product. Contact Rohde & Schwarz customer support at https://www.rohde-schwarz.com/support.

Lifting and carrying the product

The maximum weight of the product is provided in the specifications document. You can lift or carry the product by yourself, if you can manage the weight on your own. Alternatively, you can use lifting or transporting equipment. Follow the instructions provided by the equipment manufacturer.

Choosing the operating site

Only use the product indoors. The product casing is not waterproof. Water that enters can electrically connect the casing with live parts, which can lead to electric shock, serious personal injury or death if you touch the casing.

If Rohde & Schwarz provides accessories designed for outdoor use of your product, e.g. a protective cover, you can use the product outdoors.

You can operate the product up to an altitude of 2000 m above sea level. If a higher altitude is permissible, the value is provided in the specifications document. The product is suitable for pollution degree 2 environments where nonconductive contamination can occur. For more information on environmental conditions such as ambient temperature and humidity, see the specifications document.

Setting up the product

Always place the product on a stable, flat and level surface with the bottom of the product facing down. If the product is designed for different positions, secure the product so that it cannot fall over.

If the product has foldable feet, always fold the feet completely in or out to ensure stability. The feet can collapse if they are not folded out completely or if the product is moved without lifting it. The foldable feet are designed to carry the weight of the product, but not an extra load.

If stacking is possible, keep in mind that a stack of products can fall over and cause injury.

If you mount products in a rack, ensure that the rack has sufficient load capacity and stability. Observe the specifications of the rack manufacturer. Always install the products from the bottom shelf to the top shelf so that the rack stands securely. Secure the product so that it cannot fall off the rack.

Connecting the product

Before connecting the interfaces or measuring inputs of the product to other products or electrical circuits, make sure that the other products or electrical circuits provide

special protection against electric shock. This protection principle is referred to as SELV (safety extra-low voltage) and is based on a low voltage level and increased insulation. Exceptions are indicated by a measurement category on the product and given in the specifications document.

Connecting to power

The product is an overvoltage category II product. Connect the product to a fixed installation used to supply energy-consuming equipment such as household appliances and similar loads. Keep in mind that electrically powered products have risks, such as electric shock, fire, personal injury or even death. Replace parts that are relevant to safety only by original parts, e.g. power cables or fuses.

Take the following measures for your safety:

- Before switching on the product, ensure that the voltage and frequency indicated on the product match the available power source. If the power adapter does not adjust automatically, set the correct value and check the rating of the fuse.
- If a product has an exchangeable fuse, its type and characteristics are indicated next to the fuse holder. Before changing the fuse, switch off the product and disconnect it from the power source. How to change the fuse is described in the product documentation.
- Only use the power cable delivered with the product. It complies with country-specific safety requirements. Only insert the plug into an outlet with protective conductor terminal.
- Only use intact cables and route them carefully so that they cannot be damaged. Check the power cables regularly to ensure that they are undamaged. Also ensure that nobody can trip over loose cables.
- Only connect the product to a power source with a fuse protection of maximum 20 A.
- Ensure that you can disconnect the product from the power source at any time. Pull the power plug to disconnect the product. The power plug must be easily accessible. If the product is integrated into a system that does not meet these requirements, provide an easily accessible circuit breaker at the system level.

Cleaning the product

Use a dry, lint-free cloth to clean the product. When cleaning, keep in mind that the casing is not waterproof. Do not use liquid cleaning agents.

Meaning of safety labels

Safety labels on the product and its accessories warn against potential hazards.

	Potential hazard Read the product documentation to avoid personal injury or product damage.

	Electrical hazard Indicates live parts. Risk of electric shock, fire, personal injury or even death.

	Hot surface Do not touch. Risk of skin burns. Risk of fire.

	Protective conductor terminal Connect this terminal to a grounded external conductor or to protective ground. This connection protects you against electric shock if an electric problem occurs.

1.2 Warning messages in the documentation

A warning message points out a risk or danger that you need to be aware of. The signal word indicates the severity of the safety hazard and how likely it will occur if you do not follow the safety precautions.

WARNING

Potentially hazardous situation. Could result in death or serious injury if not avoided.

NOTICE

Potential risks of damage. Could result in damage to the supported product or to other property.

1.3 Korea certification class A



이 기기는 업무용(A급) 전자파 적합기기로서 판매자 또는 사용자는 이 점을 주의하시기 바라며, 가정외의 지역에서 사용하는 것을 목적으로 합니다.

2 Welcome to the R&S ZNB

This manual is intended to provide you with all information that is necessary for setup, manual and remote control of the R&S ZNB.

We also invite you to find out what's new in the current revision of the software and learn how to make best use of our documentation and of the help system.

If you have any questions or comments, please contact your partners at Rohde & Schwarz and give us your feedback.

2.1 Documentation overview

This section provides an overview of the R&S ZNB user documentation. Unless specified otherwise, you find the documents at:

2.1.1 Getting started manual

Introduces the R&S ZNB and describes how to set up and start working with the product. Includes basic operations, typical measurement examples, and general information, e.g. safety instructions, etc.

A printed version is delivered with the instrument. A PDF version is available for download on the Internet.

2.1.2 User manual and help

The user manual contains the description of all instrument modes and functions. It also provides an introduction to remote control, a complete description of the remote control commands with programming examples, and information on maintenance, instrument interfaces and error messages. Includes the contents of the getting started manual.

The contents of the user manual are available as help in the R&S ZNB. The help offers quick, context-sensitive access to the complete information for the instrument and its firmware.

The user manual is also available for download or for immediate display on the Internet.

2.1.3 Service manual

Describes the performance test for checking the rated specifications, module replacement and repair, firmware update, troubleshooting and fault elimination, and contains mechanical drawings and spare part lists.

The service manual is available for registered users on the global Rohde & Schwarz information system (GLORIS):

R&S ZNB Classified Service Document

2.1.4 Instrument security procedures

Deals with security issues when working with the R&S ZNB in secure areas. It is available for download on the internet.

2.1.5 Printed safety instructions

Provides safety information in many languages. The printed document is delivered with the product.

2.1.6 Specifications document and brochure

The specifications document, also known as the data sheet, contains the technical specifications of the R&S ZNB. It also lists the firmware applications and their order numbers, and optional accessories.

The brochure provides an overview of the instrument and deals with the specific characteristics.

https://www.rohde-schwarz.com/brochure-datasheet/ZNB.

2.1.7 Release notes and open source acknowledgment (OSA)

The release notes list new features, improvements and known issues of the current software version, and describe the software installation.

The software uses several valuable open source software packages. An open source acknowledgment document provides verbatim license texts of the used open source software.

https://www.rohde-schwarz.com/firmware/ZNB.

An HTML version of the OSA is installed with the analyzer firmware. It can be opened from the main menu of the VNA's graphical user interface ("Help">"About">"Open Source Acknowledgment: Open...").

2.1.8 Application notes, application cards, white papers, etc.

These documents deal with special applications or background information on particular topics.

https://www.rohde-schwarz.com/application/ZNB

3 Getting started

Note: the following chapters are identical to those in the printed R&S ZNB Getting Started manual.

- Preparing for use....21
- Instrument tour....35
- Operating the instrument....44
- Performing measurements....70

3.1 Preparing for use

Here, you can find basic information about setting up the product for the first time.

3.1.1 Lifting and carrying

The carrying handles are designed to lift or carry the instrument. Do not apply excessive external force to the handles.

See "Lifting and carrying the product" on page 16.

3.1.2 Unpacking and checking

1. Unpack the R&S ZNB carefully.
2. Retain the original packing material. Use it when transporting or shipping the R&S ZNB later.
3. Using the delivery notes, check the equipment for completeness.
4. Check the equipment for damage.

   If the delivery is incomplete or equipment is damaged, contact Rohde & Schwarz.

3.1.3 Choosing the operating site

Specific operating conditions ensure proper operation and avoid damage to the product and connected devices. For information on environmental conditions such as ambient temperature and humidity, see the specifications document.

For safety information, see "Choosing the operating site" on page 16.

Electromagnetic compatibility classes

The electromagnetic compatibility (EMC) class indicates where you can operate the product. The EMC class of the product is given in the specifications document.

- Class B equipment is suitable for use in:
  - Residential environments
  - Environments that are directly connected to a low-voltage supply network that supplies residential buildings
- Class A equipment is intended for use in industrial environments. It can cause radio disturbances in residential environments due to possible conducted and radiated disturbances. It is therefore not suitable for class B environments.
  If class A equipment causes radio disturbances, take appropriate measures to eliminate them.

3.1.4 Setting up the product

See also:

- "Setting up the product" on page 16
- "Intended use" on page 15

3.1.4.1 Placing the product on a bench top

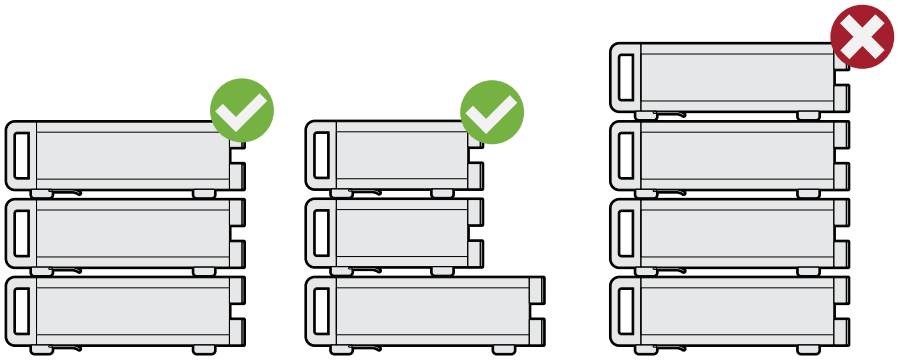
To place the product on a bench top

1. Place the product on a stable, flat and level surface. Ensure that the surface can support the weight of the product. For information on the weight, see the specifications document.
2. **CAUTION!** Foldable feet can collapse. For safety information, see "Setting up the product" on page 16.

   Always fold the feet completely in or out. With folded-out feet, do not place anything on top or underneath the product.
3. **WARNING!** A stack of products can fall over and cause injury. Never stack more than three products on top of each other. Instead, mount them in a rack.

   Stack as follows:

   - If the products have foldable feet, fold them in completely.
   - It is best if all products have the same dimensions (width and length). If the products have different dimensions, stack according to size and place the smallest product on top.
   - Do not exceed the permissible total load placed on the product at the bottom of the stack:
     - 50 kg when stacking products of identical dimensions (left figure).
     - 25 kg when stacking smaller products on top (middle figure).



Left = Stacked correctly, same dimensions
Middle = Stacked correctly, different dimensions
Right = Stacked incorrectly, too many products

4. **NOTICE!** Overheating can damage the product.
   Prevent overheating as follows:
   - Keep a minimum distance of 10 cm between the fan openings of the product and any object in the vicinity to provide sufficient airflow and ventilation.
   - Do not place the product next to heat-generating equipment such as radiators or other products.

3.1.4.2 Mounting the product in a rack

To prepare the rack

1. Observe the requirements and instructions in "Setting up the product" on page 16.
2. **NOTICE!** Insufficient airflow can cause overheating and damage the product.
   Design and implement an efficient ventilation concept for the rack.

To mount the R&S ZNB in a rack

1. Use an adapter kit to prepare the R&S ZNB for rack mounting.
   a) Order a suitable rack adapter kit.
   See the specifications document.
   b) Mount the adapter kit. Follow the assembly instructions provided with the adapter kit.
2. Lift the R&S ZNB to shelf height.
3. Grab the handles and push the R&S ZNB onto the shelf until the rack brackets fit closely to the rack.
4. Tighten all screws in the rack brackets with a tightening torque of 1.2 Nm to secure the R&S ZNB in the rack.

To unmount the R&S ZNB from a rack

1. Loosen the screws at the rack brackets.
2. Remove the R&S ZNB from the rack.
3. If placing the R&S ZNB on a bench top again, unmount the adapter kit from the R&S ZNB. Follow the instructions provided with the adapter kit.

3.1.5 Considerations for test setup

Cable selection and electromagnetic interference (EMI)

Electromagnetic interference (EMI) can affect the measurement results.

To suppress electromagnetic radiation during operation:

- Use high-quality shielded cables, for example, double-shielded RF and LAN cables.
- Always terminate open cable ends.
- Ensure that connected external devices comply with EMC regulations.

Regarding length and quality, the following requirements have to be met for cables that are directly connected to the R&S ZNB:

Table 3-1: Cable Requirements

Cable Type (Connector)	Requirement
RF cables (PORT 1, ..., PORT N)	Double shielded
BNC cables (various)	Double shielded
DB-25 (USER PORT)	Double shielded
Digital I/O (Direct Control	R&S ZN-B121, order no. 1323.9290.00
GPIB	Standard cable
Handler I/O	Standard cable
RFFE/GPIO	R&S ZN-Z25 (order no. 1334.3424.02)
DisplayPort (Monitor)	Standard cable
HDMI (Monitor)	Standard cable
DVI-D (Monitor)	2 ferrite cores
LAN	At least CAT6, S/FTP
PCIe	Standard cable
USB	Standard cables, length ≤ 3m

Signal input and output levels

Information on signal levels is provided in the specifications document. Keep the signal levels within the specified ranges to avoid damage to the product and connected devices.

Preventing electrostatic discharge (ESD)

Electrostatic discharge is most likely to occur when you connect or disconnect a DUT.

► **NOTICE!** Electrostatic discharge can damage the electronic components of the product and the device under test (DUT).

Ground yourself to prevent electrostatic discharge damage:

a) Use a wrist strap and cord to connect yourself to ground.
b) Use a conductive floor mat and heel strap combination.

During operation, if the firmware observes a serious unexpected disturbance (e.g. due to ESD), it resets all hardware components to ensure proper instrument functioning. It then restores the user settings to the state before the disturbance and indicates the foregone hardware reset by an "Hardware communication problem [...]" information popup.

3.1.6 Connecting the analyzer to the AC supply

For safety information, see "Connecting to power" on page 17.

The network analyzer is automatically adapted to the AC supply voltage, which must be in the range of 100 V to 240 V at 50 Hz to 60 Hz. A line frequency of 400 Hz is also supported.

The AC power connector is located in the upper part of the rear panel (see Section 3.2.2, "Rear panel", on page 41).

1. Plug the AC power cable into the AC power connector on the rear panel of the product. Only use the AC power cable delivered with the product.
2. Plug the AC power cable into a power outlet with ground contact.

   The required ratings are listed next to the AC power connector and in the specifications document.
3. If necessary, also ground the product using the grounding terminal ⏚.

The maximum power consumption and the typical power consumption of the individual analyzer models are listed in the specifications document.

3.1.7 Switching the instrument on and off

To switch on the R&S ZNB

1. Switch the AC power switch to position **I** (On).
   The AC power switch is located in the upper part of the rear panel, together with the mains connector and the fuse drawer; see Section 3.2.2, "Rear panel", on page 41.

   After power-on, the analyzer automatically goes to standby or ready state, depending on the state of the standby toggle key at the front panel when the instrument was switched off last time.

2. If necessary, press the standby toggle key on the front panel to switch the instrument to the ready state.

   See Section 3.1.8, "Standby and ready state", on page 26.

   The instrument automatically performs a system check, boots the Windows® operating system and then starts the vector network analyzer (VNA) application. If it was terminated regularly, the VNA application restores all recall sets and instrument settings of the previous analyzer session.

To shut down the instrument

1. Press the standby toggle key.

   Pressing the standby toggle key causes the instrument to save all loaded recall sets, to close the VNA application, to shut down Windows®, and to go to standby state (see Section 3.1.8, "Standby and ready state", on page 26).
   Of course, you can also perform these steps manually, like in any Windows session.

2. If desired, set the AC power switch to position **O** (Off).

To disconnect from power

Prerequisite: The R&S ZNB is in standby state.

1. **NOTICE!** Risk of data loss. If you disconnect the product from power when it is in the ready state, you can lose settings and data. Shut it down first.

   Set the switch on the power supply to position [0].

   The LED of the standby toggle key key is switched off.

2. Disconnect the R&S ZNB from the power source.

3.1.8 Standby and ready state



The standby toggle key is located in the bottom-left corner of the front panel. In standby state, the right, amber LED is on, in ready state, the left, green LED is on.

The standby power only supplies the power switch circuits and the optional high precision quartz (R&S ZNB-B4). In this state, it is safe to switch the AC power off and disconnect the instrument from the power supply. In ready state, all modules are power-supplied. When switched to ready state, the analyzer initiates its startup procedure.

Observe the instructions for startup and shutdown in Section 3.1.7, "Switching the instrument on and off", on page 26.

3.1.9 Windows operating system

The analyzer is equipped with a Windows® operating system, which has been configured according to the instrument's features and needs. Changes in the system configuration can be necessary to:

- Establish a LAN connection.
- Customize the properties of the external accessories connected to the analyzer.
- Call up additional software tools.

All necessary settings can be accessed from the Windows "Start" menu, in particular from the "Control Panel". To open the "Start" menu, press the Windows key in the toolbar or on an external keyboard.

NOTICE

Modifications of the operating system

The operating system is adapted to the network analyzer. To avoid impairment of instrument functions, only change the settings described in this manual.

Only execute programs that are authorized by Rohde & Schwarz for use on the instrument. Only modify installed software with updates released by Rohde & Schwarz.

User accounts and password protection

The analyzer uses a user name and password as credentials for remote access. Two user accounts with different levels of access are available on the instrument:

- "instrument" is the default account with standard rights to change system settings. Use this account for normal operation of the analyzer.
- "Admin" is the account for administering the operating system. This account is required, for instance, if you wish to install programs on the analyzer.

In the factory configuration, "894129" is preset as a password for both users. To protect the analyzer from unauthorized access, it is recommended to change the preset passwords.

To switch from one user account to another, log off from Windows and then log on again. The "switch user" functionality is disabled on the R&S ZNB.

Security updates and patches

Microsoft regularly creates security updates and other patches to protect Windows-based operating systems. They are released through the Microsoft Update website and associated update server. Update instruments using Windows regularly, especially instruments that connect to a network.

The analyzer is equipped with a Windows 10 IoT Enterprise LTSB/LTSC operating system, which is the embedded version of Windows 10 with long-term support for Windows security updates and patches. The firmware is tested with the following OS versions:

- Windows 10 IoT Enterprise 2016 LTSB
- Windows 10 IoT Enterprise 2019 LTSC
- Windows 10 IoT Enterprise 2021 LTSC

Firewall settings

A firewall protects an instrument by preventing unauthorized users from gaining access to it through a network. Rohde & Schwarz highly recommends using the firewall on your instrument. Rohde & Schwarz instruments are shipped with the Windows firewall enabled. All ports and connections for remote control are enabled.

Note that changing firewall settings requires administrator rights.

Virus protection

Take appropriate steps to protect your instruments from infection. Use strong firewall settings and scan any removable storage device used with a Rohde & Schwarz instrument regularly. It is also recommended that you install anti-virus software on the instrument. Rohde & Schwarz does NOT recommend running anti-virus software in the background ("on-access" mode) on Windows-based instruments, due to potentially degrading instrument performance. However, Rohde & Schwarz does recommend running it during non-critical hours.

For details and recommendations, see the Rohde & Schwarz white paper 1EF96: Malware Protection Windows 10.

3.1.10 Minimizing the VNA application

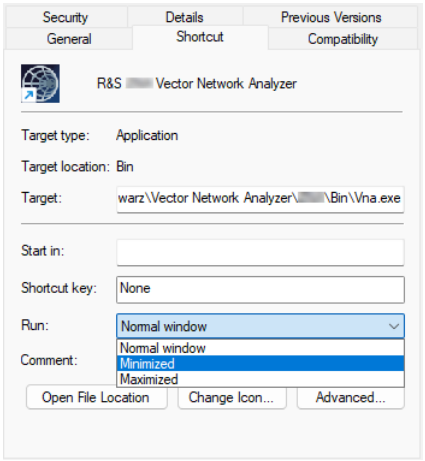
With a minimized VNA application, you can access your analyzer's Windows® desktop or run other applications.

To exit the default full-screen mode of the VNA application, deselect System – [Display] > "View Bar" > "Title Bar Task Bar". Then use the standard Windows® title bar functions to minimize/maximize/close the application window.

To start the VNA application with a minimized window

1. Right-click the Vector Network Analyzer shortcut icon on the Windows desktop.
2. Open the "Properties" dialog.

3. In the "Shortcut" tab, select "Run: Minimized".



A software update restores the original shortcut properties.

3.1.11 Connecting external accessories

The analyzer's standard PC interfaces (Monitor, USB, LAN) can be used to connect various accessories:

- An external monitor expands/displays the Windows® desktop, which is, by default, covered by the vector network analyzer (VNA) application window in full-screen mode.
- External keyboard and mouse simplify local control, in particular manual (GUI) operation of the VNA application.
- A printer can be used to create hardcopies of the measurement diagrams and traces from within the VNA application.
- A LAN connection can be established to access the analyzer's mass storage or control the analyzer from an external PC.
- Instruments equipped with the latest controller board can also be remote-controlled via USB.

3.1.11.1 Connecting a monitor

Connect a standard monitor to one of the monitor connectors on the rear panel of the instrument.



DisplayPort (recent controller boards only)



HDMI™ (2nd generation R&S ZNB only)



DVI-D (1st generation R&S ZNB only)

At the graphical user interface of the VNA, activate "System" – "Display" > "View Bar" > "Hard Key Panel" to add the (virtual) "Hard Key Panel" to the application window.

3.1.11.2 Connecting a keyboard

A keyboard can be connected to any of the USB connectors. After being auto-detected by the operating system, it can safely be disconnected and reconnected even during measurements.

Keyboard configuration

The default input language is English – US. Select "Settings" > "Time & language" > "Region"/"Language" from the Windows® Start menu to manage languages and keyboards.



To access Windows®, use the ■ button in the toolbar of the application window.

3.1.11.3 Connecting a mouse

A USB mouse can be connected to any of the USB type A connectors. After being auto-detected by the operating system, it can safely be disconnected and reconnected even during measurements.

Mouse configuration

Select "Settings" > "Devices" > "Mouse" from the Windows® "Start" menu to configure the mouse properties.



To access Windows®, use the ■ button in the toolbar of the application window.

3.1.11.4 Connecting a printer

A printer can be connected to any of the USB connectors. After successful installation, it can safely be disconnected and reconnected even during measurements.

Before printing (System – [Print]), the analyzer checks whether a printer is connected and turned on and whether the appropriate printer driver is installed.

Printer driver installation

If necessary, the printer driver installation is initiated using the operating system's "Add a printer or scanner". The control is self-explanatory. A printer driver must be installed only once.

A great variety of printer drivers are available on the analyzer. To obtain the complete list, select "Settings" > "Devices" > "Printers & scanners" from the Windows® "Start" menu.

To access Windows®, use the ■ button in the toolbar of the application window.

You can load updated and improved driver versions or new drivers from an installation disk, USB memory stick or another external storage medium. Alternatively, if the analyzer is integrated in a network, you can install driver data stored in a network directory. In either case, use the "Add a printer or scanner" control to complete the installation.

Printer configuration

Use the "Printer Setup" dialog of the firmware (System – [Print] > "Print ►") or the Windows® printer management to configure the printer properties and printing preferences.

3.1.11.5 Connecting a USB cable for remote control

Instruments equipped with the latest controller board can also be remote-controlled via USB. To prepare for remote control operation, connect a suitable USB 2.0 or 3.0 cable to the type B "USB Device" port on the rear panel of the instrument. With direct connection to a host device, a connecting cable A-B (plug type A onto plug type B) must be used.

For more information on remote control, refer to Section 6.1, "Introduction to remote control", on page 868.

3.1.12 Remote operation in a LAN

A LAN connection is used to integrate the analyzer into a home/company network. LAN connectivity offers several applications, e.g.:

- Transfer data between a controller and the analyzer, e.g. to run a remote control program.
- Control the measurement from a remote computer using Remote Desktop or a similar application.
- Use external network devices (e.g. printers).

The analyzer uses a user name and password as credentials for remote access; see Section 3.1.9, "Windows operating system", on page 27 for details. To protect the analyzer from unauthorized access, it is recommended to change the factory setting.

3.1.12.1 Assigning an IP address

Depending on the network capacities, the TCP/IP address information for the analyzer can be obtained in different ways.

- If the network supports dynamic TCP/IP configuration using the Dynamic Host Configuration Protocol (DHCP), all address information can be assigned automatically.
- If the network does not support DHCP, or if the analyzer is set to use an alternate TCP/IP configuration, the addresses must be set manually.

By default, the analyzer is configured to use a dynamic TCP/IP configuration and obtain all address information automatically. Hence it is safe to establish a physical connection to the LAN without any previous analyzer configuration.

NOTICE

Manual TCP/IP configuration

If your network does not support DHCP, or if you choose to disable dynamic TCP/IP configuration, you must assign valid address information **before** you connect the analyzer to the LAN. Contact your network administrator to obtain a valid IP address, because connection errors can affect the entire network.

Admin account

You need administrator rights to change the TCP/IP configuration. See Section 3.1.9, "Windows operating system", on page 27 for details.

To enter the TCP/IP address information manually

1. Obtain the IP address and subnet mask for the analyzer and the IP address for the local default gateway from your network administrator. If necessary, also obtain the name of your DNS domain and the IP addresses of the DNS and WINS servers on your network.
2. Tap the Windows icon in the toolbar, or press the Windows key in the System keypad (or on an external keyboard) to access Windows®.
3. Open the "Control Panel" and select "Network and Internet" > "Network and Sharing Center" > "Change adapter settings" for the Ethernet adapter connection.
4. Pass the Windows user account control, if necessary:
   - If your current account is an administrator account, select "Yes".
   - If your account is an account with standard user rights, enter the password of the administrator account and select "Yes".

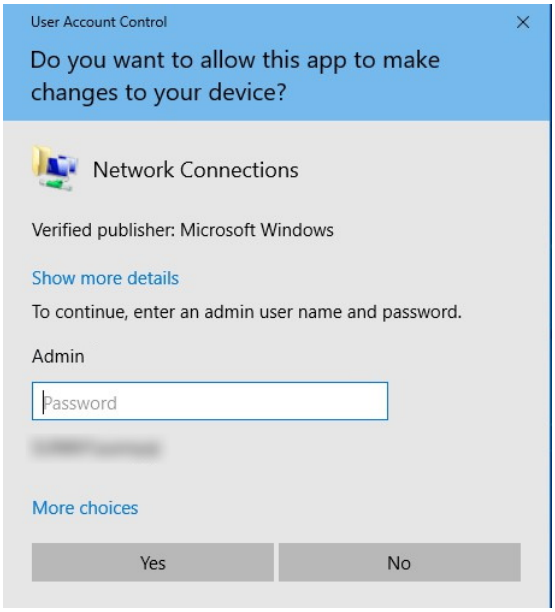
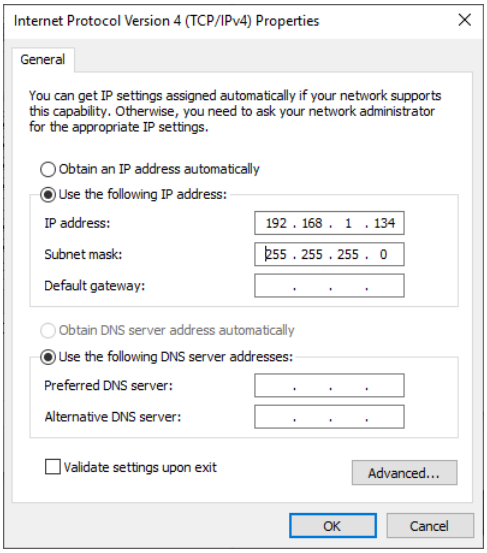


Figure 3-1: User Account Control dialog

5. In the "Ethernet Properties" dialog, select "Internet Protocol Version 4 (TCP/IPv4)" and press "Properties".
6. In the properties dialog, enter the IP address information.



3.1.12.2 Using computer names

In a LAN that uses a DNS server (Domain Name System server), each PC or instrument connected in the LAN can be accessed via an unambiguous computer name instead of the IP address. The DNS server translates the host name to the IP address, which is especially useful when a DHCP server is used, as a new IP address can be assigned each time the instrument is restarted.

Each instrument is delivered with an assigned computer name, but this name can be changed. The default instrument name is a case-insensitive string with the following syntax:

ZNB<4|8|20|26|40|43>-<variant>-<serial_number>, ZNB40-N-<serial_number>

The serial number can be found on the rear panel of the instrument. It is the third part of the device ID printed on the barcode sticker:



ZNB40-84-102030, for example

3.1.12.3 Remote Desktop connection

Remote Desktop is a Windows® application that you can use to access and control the analyzer from a remote computer through a LAN connection. While the measurement is running, the analyzer screen contents are displayed on the remote computer, and Remote Desktop provides access to all applications, files, and network resources of the analyzer.

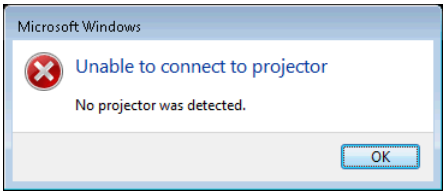
At the R&S ZNB, by default remote connections are enabled using a local group policy and remote access is granted to users instrument and administrator.

For detailed information about Remote Desktop and the connection, refer to the Windows® Help ("Windows Start Menu" > "Get Help").

To establish a remote desktop connection, proceed as follows:

1. Connect a remote Windows PC and your VNA to the LAN and make sure that an IP connection can be established.
2. At the remote Windows PC, open a remote desktop connection (type *Remote Desktop Connection* at the Windows Start/Search menu and hit enter).
3. In the "Remote Desktop Connection" dialog, enter the VNA's computer name or IP address and select "Connect"

If no external monitor is connected at the time you log in to the instrument via remote desktop, Windows® possibly displays the following error message:



This message can be safely ignored.

3.1.12.4 Windows® firewall settings

A firewall protects an instrument by preventing unauthorized users from gaining access to it through a network. In the default configuration of the R&S ZNB, the firewall is enabled. A remote desktop connection does not require any changes in the firewall settings.

Admin account

You need administrator rights to change the firewall settings. See Section 3.1.9, "Windows operating system", on page 27 for details.

Some actions require a different firewall configuration, e.g.:

- To transfer data with other hosts in the LAN, you have to allow "File and Printer Sharing".

To change the firewall settings, proceed as follows:

1. Access the operating system by pressing the Windows® key on the external keyboard. Open the "Control Panel".
2. Select "Windows Defender Firewall"
   - Select "Allow an app or feature through Windows Defender Firewall" to enable "File and Printer Sharing".
   - Select "Turn Windows Firewall on or off" to enable or disable the firewall.

Confirm the user account control message to allow the desired changes (see Figure 3-1). For detailed information about the firewall, refer to the Windows® Help.

3.2 Instrument tour

This section gives an overview of the control elements and connectors of the R&S ZNB and gives all information that is necessary to put the instrument into operation and connect external devices.

The meanings of the labels on the product are described in "Meaning of safety labels" on page 17.

3.2.1 Front panel

The front panel of the instrument consists of the touchscreen with the diagrams and softtool panels (left side), the hardkey area (right side) and the test port area below. Brief explanations on the controls and connectors, the hardkey area and the rear panel can be found in the next sections.

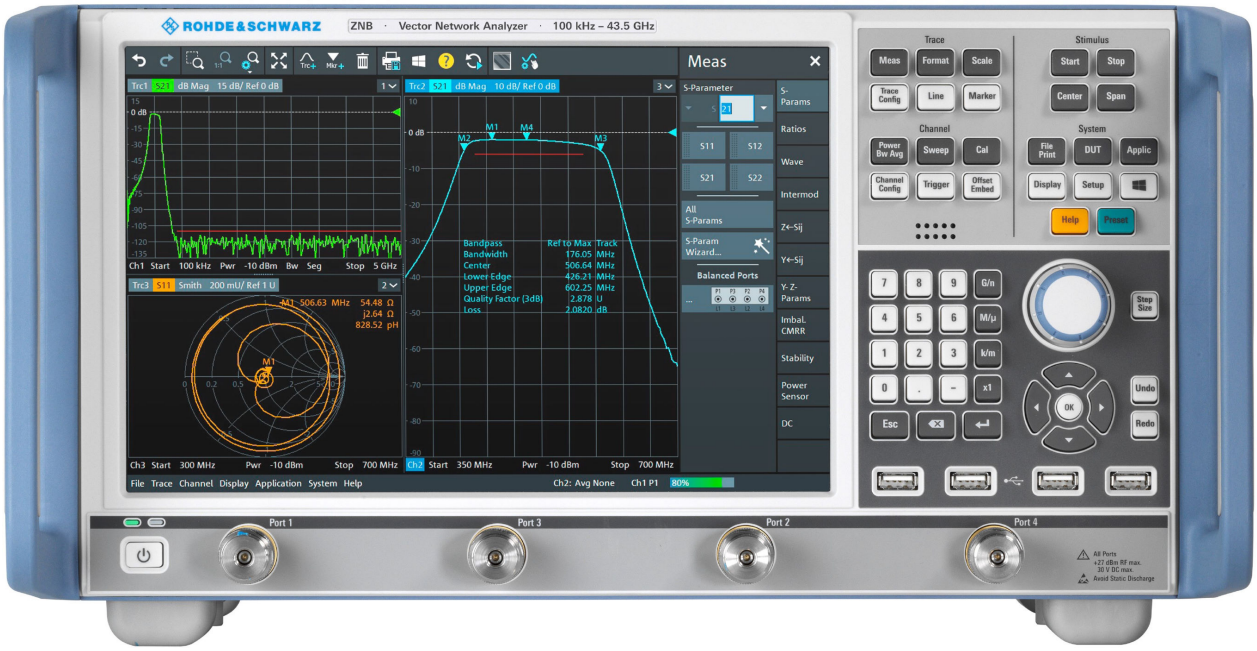


Figure 3-2: R&S ZNB43 with four ports

3.2.1.1 Touchscreen

The analyzer is equipped with a 12.1" diagonal WXGA, 18-bit color LCD touchscreen. The touchscreen presents all measurement results, mostly in the form of diagrams. Besides, all instrument functions can be accessed and operated by tapping the control elements on the touchscreen. For an introduction to touchscreen operation, refer to Section 3.3.1, "Manual operation", on page 45.

The following sections contain further useful information about manual control of the instrument.

- Refer to the other subsections of Section 3.3, "Operating the instrument", on page 44 to learn how to handle traces and diagrams, and how to use menus, keys and softtools.
- Refer to Section 4.2.1, "Display elements of a diagram", on page 93 to obtain information about the results in the diagram.

- Refer to the section Section 5.16, "Display softtool", on page 795 to learn how to customize the screen.
- Refer to the specifications document for the technical specifications of the display.

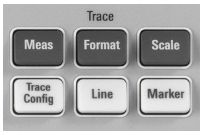
Screen saver

The screen saver function of the operating system can be used to switch off the display if the analyzer receives no command for a selectable period of time. The display is switched on again if any front panel key is pressed.

To enable the screen saver, access the Windows® operating system (e.g. by pressing the Windows key in the System keypad) and select "Control Panel > Power Options > Change Plan Settings > Turn off the display".

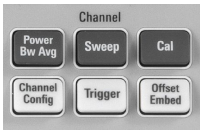
3.2.1.2 Function keys

Most of the keys in the Trace, Channel, Stimulus, and System keypads call up a related softtool of the analyzer GUI. Every softtool provides access to a group of related measurement settings.



The Trace keys give access to all trace settings, to the limit check settings, and to the marker functions including marker search.

- The [Meas] settings select the measured and displayed quantity.
- The [Format] settings define how measured data (traces) are presented.
- The [Scale] settings define how traces are scaled.
- The [Trace Config] settings store traces to the memory and perform mathematical operations on traces.
- The [Line] settings define limits for the measurement results, visualize them in the diagrams and activate/deactivate the limit check.
- The [Marker] settings position markers on a trace, configure their properties and select the format of the numerical readout. Markers can also be used to locate specific points on the trace, define the sweep range, and scale the diagram.



The Channel keys give access to the hardware-related (channel) settings.

- The [Pwr Bw Avg] settings define the power of the internal signal source, the IF bandwidth, and the sweep average.
- The [Sweep] settings define the scope of measurement, including the sweep type and the number of measured sweeps.
- [Cal] provides all functions that are necessary to perform a system error correction (calibration).
- [Channel Config] provides functions for channel management.
- The [Trigger] settings control the start of the measurement sequence.
- [Offset Embed] provides a selection of length offset parameters to shift the measurement plane.



The Stimulus keys ([Start], [Stop], [Center], [Span]) define the sweep range, depending on the sweep type.



Figure 3-3: R&S ZNB system keys (left: 1st gen | right: 2nd gen)

The System keys provide general system settings.

- [File] provides standard Windows® functions used to create, save, recall or print recall sets, to copy the active screen and to shut down the application.
- The [Print] settings control an external printer that is used to print a hardcopy of the current recall set.
- [File Print] is available on 2nd generation R&S ZNB only. Gives access to the [FILE] and [PRINT] settings.
- [DUT] opens a wizard that allows you to set up a measurement by specifying the properties of the DUT (2nd generation R&S ZNB only)
- [Applic] gives access to external software tools and optional extensions of the analyzer firmware.
- [Display] gives access to all display settings and to the functions which activate, modify and arrange different diagrams.
- [Setup] provides general system settings which are not restricted to a particular recall set.
- The Windows® key opens the startup menu of the Windows® operating system from where you can perform system configurations and call up additional software utilities.
- [Help] opens the context-sensitive help system of the analyzer.
- [Preset] performs a preset of the instrument settings.

3.2.1.3 Data entry keys



The keys in the data entry keypad are used to enter numbers, units, and characters. The data entry keys are only enabled while the cursor is placed on a data input field in a dialog or in the Help navigation pane.

- The keys 0 to 9 enter the corresponding numbers.
- The function of the "." and "–" keys depends on the data type of the active input field:
  - In numeric input fields, the keys enter the decimal point and a negative sign for the entered numeric value. Multiple entries have not effect.

  - In character input fields, the keys enter a dot and a hyphen, respectively. Both entries can be repeated as often as desired.
- The function of the four unit keys depends on the data type of the active input field; see Section 3.3.6, "Entering data", on page 61.
  - In numeric input fields, the [G/n] and [M/µ], keys multiply the entered value with factors of $10^{\pm 9}$, $10^{\pm 6}$, $10^{\pm 3}$, respectively, and add the appropriate physical unit.
  - In character input fields, the [G/n], [M/µ], [k/m] keys enter the letters G, M, K, respectively.
    [x1] is equivalent to [↵] (Enter) and confirms the previous entry.
- [Esc] is used to:
  - Cancel entries / close dialogs without activating the entries made (equivalent to the "Close" button).
  - Close the help system.
- [↵] (Enter) is used to:
  - Activate the selected active control element, e.g. a button in a dialog or a link in the "Contents" page of the Help system.
  - Confirm selections and entries made and close dialogs.
    Same functionality as the [OK] button in the navigation area.
  - Compress or expand menus or the help system's table of contents
- [←] (Backspace) deletes the last character before the cursor position or the selected character sequence or numeric value.

3.2.1.4 Rotary knob



The rotary knob increases and decreases numerical values, scrolls within lists, activates controls and confirms entries.

- Turning the rotary knob to the left or right is equivalent to pressing the [⇧] (up) or [⇩] (down) navigation key multiple times.
- Pressing the rotary knob is equivalent to pressing the [↵] (Enter) data entry key or the [OK] navigation key.

[Step Size] opens an input box to select the steps (in units of the current physical parameter) between two consecutive numerical values. The step size is also valid for value changes using the [⇧] (up) and [⇩] (down) keys. See also Section 3.3.6.2, "Using the numeric editor", on page 63.

3.2.1.5 Navigation keys



Use the keys in the navigation keypad to navigate within the analyzer GUI and help system.

With [⇧] (up) and [⇩] (down) you can:

- Scroll up and down in lists, e.g. among menu items, in a list of keywords, or in the help system's table of contents or index
  [⇧] (up) becomes inactive as soon as the beginning of the list is reached. [⇩] (down) becomes inactive as soon as the end of the list is reached.

- Navigate between table rows in diagrams
  Then press [OK] to toggle between navigation and data entry mode.
- Increase and decrease numerical input values

Instead of pressing [⇧] (up) or [⇩] (down) you can also turn the rotary knob to the right or left, respectively.

The [⇦] (left) and [⇨] (right) keys are used to:

- Move the cursor to the left or right within input fields
- Navigate between table columns in diagrams.
  (Then press [OK] to toggle between navigation and data entry mode.)
- Compress or expand menus or the help system's table of contents
- Move the highlighted item in the menu bar of the active application

[OK] is equivalent to pressing the rotary knob or the [↵] (Enter) key in the data entry keypad.

[Undo] reverses the last action, if possible. [Redo] reverses the action of the [Undo] button.

3.2.1.6 Standby key



The standby toggle switch is located in the bottom-left corner of the front panel.

The key serves two main purposes:

- Toggle between standby and ready state; see Section 3.1.8, "Standby and ready state", on page 26.
- Shut down the instrument; see Section 3.1.7, "Switching the instrument on and off", on page 26.

3.2.1.7 Front panel connectors

The test ports and four USB connectors are located on the front panel of the R&S ZNB.

Test ports



Numbered connectors:

- Type N female for R&S ZNB4 and R&S ZNB8
- 3.5 mm male for R&S ZNB20
- 2.92 mm (K) male for other R&S ZNB models

The test ports serve as outputs for the RF stimulus signal and as inputs for the measured RF signals from the DUT (response signals).

- With a single test port, it is possible to generate a stimulus signal and measure the response signal in reflection. For a measurement example, refer to Section 3.4.2, "Reflection S-parameter measurement", on page 77.
- With more than one test port, it is possible to perform full two-port, 3-port, ... , or n-port measurements; see Section 4.3.1, "S-parameters", on page 117.

In the standard configuration, all test ports are supplied by a common source. Four-port instruments are available with an optional second source.

- Use a torque wrench when screwing RF cables on the test port connectors.
- See also Section 3.1.5, "Considerations for test setup", on page 24.

USB connectors

Four high-speed Universal Serial Bus connectors of type A.



The USB ports can be used to connect:

- External PC accessories such as mouse or other pointing devices, a keyboard, printer or external storage device (USB flash drive, CD-ROM drive etc.).
- External measurement equipment such as calibration units, power meters, signal generators or switch matrices.

3.2.2 Rear panel

This section gives an overview of the rear panel controls and connectors of the network analyzer.

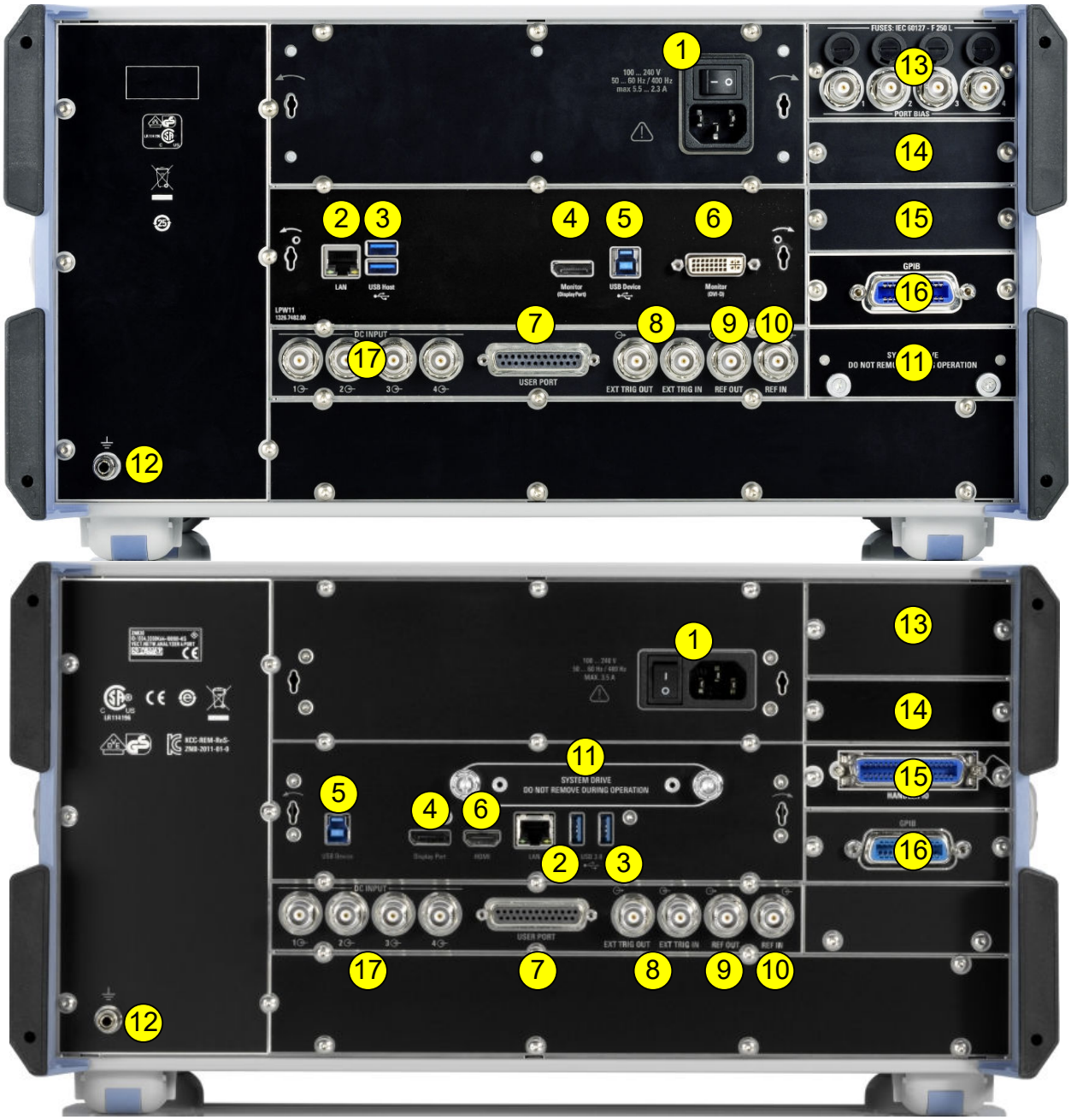


Figure 3-4: R&S ZNB rear view

top = 1st generation R&S ZNB (with latest CPU board LPW11)
bottom = 2nd generation R&S ZNB (order nos. 1334.3330.xx)

Table 3-2: Rear panel connectors available on all instruments

Index	Label	Description
1	(Power I/O)	Power on / off switch, see Section 3.1.7, "Switching the instrument on and off", on page 26
2	LAN	RJ-45 connector to integrate the instrument to a Local Area Network, primarily for remote control purposes. See Section 3.1.12, "Remote operation in a LAN", on page 31.
3	USB / USB Host	Two additional type A USB 3.0 host connectors; similar functionality as the USB connectors on the front panel (see "USB connectors" on page 41).
4	Display Port	External monitor connector (DisplayPort); see Section 3.1.11.1, "Connecting a monitor", on page 29. (n.a. for 1st gen R&S ZNB with CPU board before LPW11)
5	USB Device	Type B USB 3.0 device connector for remote control of the instrument (see Section 3.1.11.5, "Connecting a USB cable for remote control", on page 31)
6	Monitor	External monitor connector • DVI-D for 1st generation R&S ZNB • HDMI™ for 2nd generation R&S ZNB See Section 3.1.11.1, "Connecting a monitor", on page 29.
7	USER PORT	25-pin D-Sub connector used as an input and output for low-voltage (3.3 V) TTL control signals See Section 13.3.1.1, "USER PORT", on page 1702.
8	Trigger Out, Trigger In	Two BNC connectors for 5 V TTL external trigger signals See Section 5.10.3, "Trigger In tab", on page 552 and Section 5.10.4, "Trigger Out tab", on page 557.
9	REF OUT	BNC output for the internal reference frequency of the analyzer. Use this connector to synchronize other instruments to the analyzer.
10	REF IN	BNC input for an external reference frequency. Use this connector to synchronize the analyzer to another device. See Section 5.17.2, "Freq. Ref. tab", on page 841.
11	SYSTEM DRIVE	Contains the removable system drive of the instrument, containing all software (including the operating system and the VNA application) and data. Do not remove the system drive during operation. Option R&S ZNB-B19 provides an additional removable system drive (including operating system and firmware). See Section 4.7.21, "Additional removable system drive", on page 262.
12	(Ground connector)	The ground connector provides the ground of the analyzer's supply voltage. Use this connector for ESD protection; see "Preventing electrostatic discharge (ESD)" on page 25.

Table 3-3: Optional rear panel connectors

Index	Label	Description
13	Bias Tees	Two or four additional BNC inputs labeled PORT BIAS 1 ... 4 (for two-port or four-port analyzers). The inputs can be used to apply external DC voltages (biases) to the test ports. • Not available for 2nd generation R&S ZNB20 and for 1st generation R&S ZNB40 that are equipped with the new reflectometer board (order no. 1317.8015.03) • Optional for R&S ZNB4|8 • Always equipped for other R&S ZNB models • "Bias Tees", providing two or four additional BNC inputs labeled PORT BIAS 1 ... 4 (for two-port or four-port analyzers). The inputs can be used to apply external DC voltages (biases) to the test ports. See Section 4.7.14, "Bias tees", on page 258. **Note** that there is no additional HW or SW switch for the bias tees, i.e. the voltage is applied to the RF port as soon as you connect the voltage source to the corresponding PORT BIAS connector. For fuse replacement, refer to Section 12.1, "Replacing fuses", on page 1698.
13	**or** internal RFFE - GPIO Interface	R&S ZN-B15 "RFFE - GPIO Interface" (**1st generation R&S ZNB only**) 25-pin female connector, providing: • 2 independent RF Front-End (RFFE) interfaces according to the MIPI® Alliance "System Power Management Interface Specification". • 10 General Purpose Input/Output (GPIO) pins. See Section 4.7.20, "RFFE GPIO interface", on page 260.
14	Device Control	Option R&S ZNB-B12 "Device Control" provides a PCIe and a Direct Control connector. See Section 4.7.18, "Device Control", on page 259. The Direct Control interface enables direct connections between the VNA measurement bus and one or more extension devices, such as: • An external RFFE GPIO interface R&S ZN-Z15. See Section 4.7.20, "RFFE GPIO interface", on page 260. • Switch matrices R&S ZN-Z8x.
15	HANDLER IO	Option R&S ZN-B14, Handler I/O (Universal Interface), providing a Centronics 36 input/output connector. See Section 4.7.19, "Handler I/O (universal interface)", on page 260.
16	GPIB	Option R&S ZNB-B10 provides a GPIB bus connector according to standard IEEE 488 / IEC 625. See Section 13.3.3, "GPIB interface", on page 1704.
17	DC INPUT	Option R&S ZNB-B81 "DC Inputs" provides four BNC inputs for DC measurements (adjustable to different voltage ranges). SeeSection 5.2.1.10, "DC tab", on page 319.

Match signals with 50 Ω to comply with EMC directives. See also Section 3.1.5, "Considerations for test setup", on page 24.

3.3 Operating the instrument

The following sections describe the basics of manual operation, i.e. how to access instrument functions and settings via the analyzer GUI. Manual operation is particularly useful for getting to know the instrument and for trouble shooting.

Manual and remote control of the instrument

Manual control of the R&S ZNB is possible either via its touchscreen and front panel keys, via locally connected keyboard and mouse (see Section 3.1.11, "Connecting external accessories", on page 29), or via Remote Desktop (see also Section 3.1.12, "Remote operation in a LAN", on page 31).

Alternatively, it can be remote-controlled via USB (latest controller board only), GPIB or LAN.

Manual operation and remote control are described to their full extent in the GUI reference and Command reference sections, respectively. GUI functions and their related remote commands are linked bidirectionally. Background information is provided in the Concepts and features section.

3.3.1 Manual operation

The analyzer functions are accessible via several tabbed softtools, each presenting related functions and settings. See Section 3.3.2.4, "Softtools", on page 51.

The function keys on the front panel or the on-screen "Hard Key Panel" open the most frequently used softtools.

Manual operation via function keys and softtools provides touch-friendly access to the instrument functions and settings, avoiding complicated menu structures and long operating sequences. In general, this approach is recommended. However, sometimes the toolbar or an object's context menu can offer a shortcut. The menu bar provides access to all instrument functions and settings.



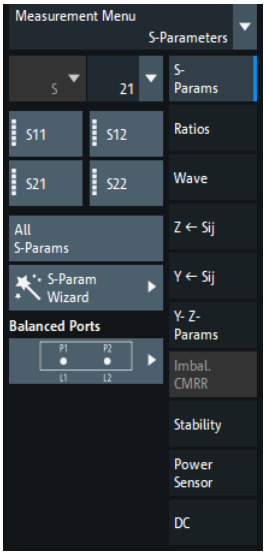
Figure 3-5: Function Keys

Using the front panel

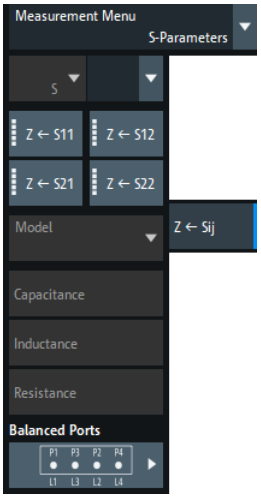
To access an instrument function:

1. Press a (virtual) key, e.g. the [Meas] key in the Trace section.

   The corresponding softtool expands at the current docking position.



2. From the "Measurement Menu", select "S-Parameters".
3. Activate the desired softtool tab, e.g. "Z←Sij".



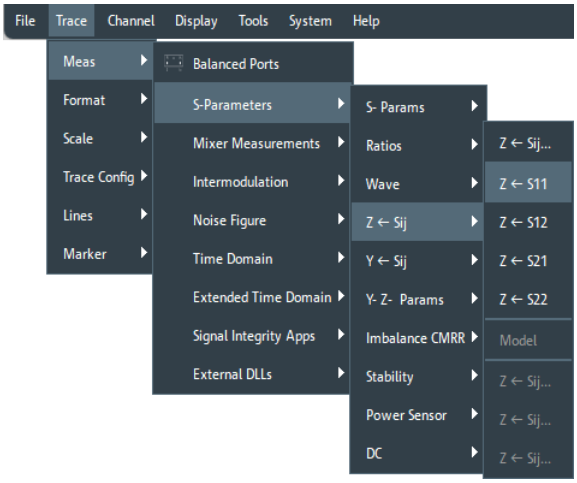
4. Select a control element, e.g. "Z←S11".

   The diagram immediately reflects your selection. The active trace shows the measurement results for the selected measured quantity.

Using the menu bar

The menu bar at the bottom of the application screen provides alternative access to **all** instrument functions. To repeat the measured quantity selection described above,

► Select "Trace" – "Meas" > "S-Parameters" > "Z←Sij" > "Z←S11".

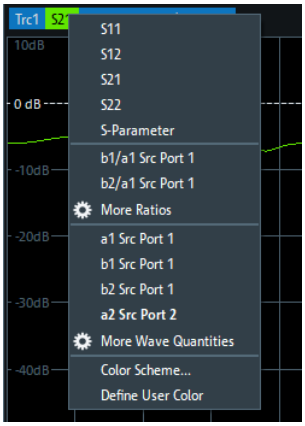


The diagram immediately reflects your selection. The active trace shows the measurement results for the selected measured quantity. At the same time, the related softtool tab is opened.

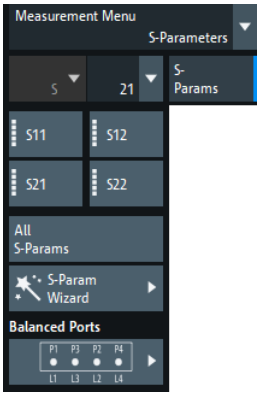
Using context menus

Context menus are another alternative for quick access to instrument settings.

1. Touch and hold (right-click) the measured quantity section in the trace info for a couple of seconds until the context menu appears.



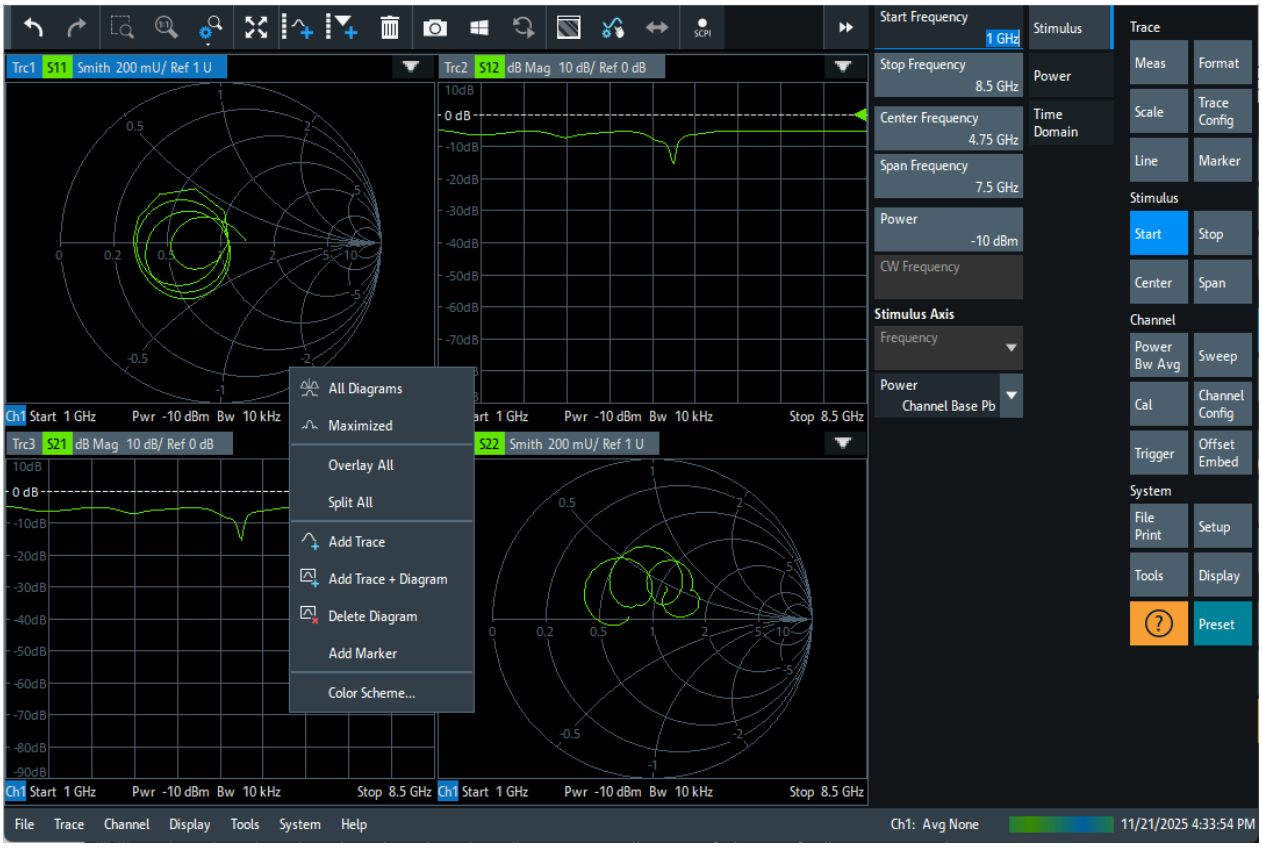
2. Select "S-Parameter" to open the "S-Params" softtool tab.



3. Select "S11".

3.3.2 Control elements of the application window

The application window of the analyzer GUI provides all control elements for the measurements and contains the diagrams for the results.



There are several alternative ways for accessing an instrument function:

- Using a function key on the (virtual) hardkey panel to open the related softtool (recommended, provides all settings)

- Using the menus and submenus of the menu bar (an alternative to the previous method)
- Using the context menus of certain display objects (for important actions in the context of this object)
- Using the icons in the toolbar above the diagram area (for frequent global actions)

These methods are described in more detail in the following sections.

Customizing the screen

The contents of the screen and the size and position of many display and control elements are not fixed. You can display or hide most elements. You can also drag and drop traces, info fields, and even the softtool panel to your preferred position; see Section 3.3.5.4, "Using drag and drop", on page 60.

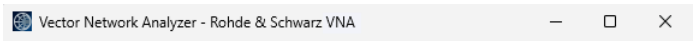
For further reference:

- Refer to Section 4.2.1, "Display elements of a diagram", on page 93 to obtain information about the results in the diagram.
- Refer to Section 5.16, "Display softtool", on page 795 and learn how to customize the screen.

3.3.2.1 Title bar

By default, the analyzer GUI is shown in full-screen mode, covering the whole screen and hiding the Windows taskbar. However, you can toggle the full-screen mode using System – [Display] > "View Bar" > "Title and Task Bar On".

If full-screen mode is switched off, the main application window of the vector network analyzer application provides a standard Windows® title bar.



3.3.2.2 Toolbar

The toolbar above the diagram area contains the most frequently used control elements of the user interface. All controls are also accessible via Softtools.



The toolbar is divided into several icon groups, separated by vertical lines.

	These icons represent the undo and redo actions that are also available via the menu bar items "System" > "Undo" / "Redo". Undo reverses the last action, redo reverses the last undo action (if possible).
	These icons control the zoom function (Trace – [Scale] > "Zoom").

	If multiple diagrams are configured, this icon toggles the "Maximize Diagram" action for the active diagram (System – [Display] > "Diagram" > "Maximize Diagram").
	These icons implement the following actions, from left to right: • Add a clone of the active trace to the active diagram (single tap; same as Trace – [Trace Config] > "Traces" > "Add Trace") or to an arbitrary/new one (drag & drop the  icon to the diagram area). • Add a new marker to the active diagram (single tap; similar to Trace – [Marker] > "Markers" > "Mkr<i>") or to an arbitrary one (drag & drop the "Mkr+" icon). • Delete: – The active trace by tapping/clicking the trash icon – An arbitrary trace by dragging and dropping its trace info field on the trash icon – All markers of a trace by dragging and dropping the trace's marker info field on the trash icon – A single marker by dragging and dropping the marker symbol on the trash icon
	These icons provide some helper functions. From left to right: • Save the current diagrams as a `BMP`, `JPEG`, `PNG`, `SVG`, or `PDF` file. By instrument default, all diagrams are included, no matter if displayed or not. Use the Printer Setup dialog to tailor the output to your needs. • Open the Windows Start menu.
	Restart the sweep in all channels. Same as Channel – [Sweep] > "Sweep Control" > "Restart Sweep"
	These icons provide the following actions: • Toggle advanced diagram area editing, which makes rearranging and/or deleting diagrams a breeze. • Toggle other measurements (except the active trace) OFF|ON
	Open the METAS Reconnection dialog This button is only visible if the measurement uncertainty analysis option is installed. It is only enabled if a METAS calibration is active.
	Open the SCPI recorder See Section 6.6, "Automating tasks with remote command scripts", on page 904.



You can toggle the visibility of the toolbar using System – [Display] > "View Bar" > "Tool Bar".

3.3.2.3 Hardkey panel

The (virtual) "Hard Key" panel provides on-screen access to the function keys of the R&S ZNB. Most of the function keys open a related softtool. For a short description, refer to section Section 3.2.1.2, "Function keys", on page 37.



The (virtual) "Hard Key" panel is particularly useful if the analyzer is controlled from an external monitor or Remote Desktop. It is hidden by default.

You can display the "Hard Key" panel using one of the following methods:

- Select System – [Display] > "View Bar" > "Hard Key Panel".
- Select "Display"> "View Bar" > "Hard Key Panel On" from the menu bar.
- Select "Hard Key" from the context menu of the softtool panel.

3.3.2.4 Softtools

Softtools display groups of related settings as a tabbed panel with a collapse/expand toggle ⏩/⏪. They can be opened via function keys on the front panel or the on-screen "Hard Key" panel, or via menu bar and context menu items.

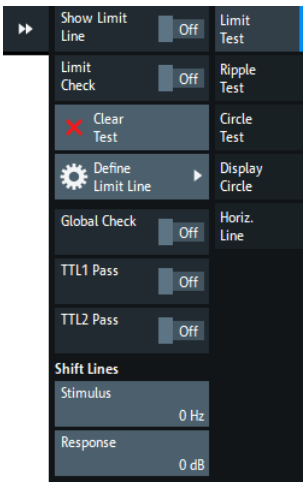


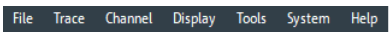
Figure 3-6: Lines softtool

Some controls on the softtool tabs allow you to read and modify settings (e.g. "Stimulus" in the screenshot above), some perform actions (e.g. "Show Limit Line"), while buttons with a ▶ symbol open additional dialogs (e.g. "Define Limit Line").

Use the context menu of the softtool panel to configure its position (right/left).

3.3.2.5 Menu bar

You can also access the analyzer functions via textual menus. The menu bar is located below the diagram area:



As in any Windows® application, menus can be controlled with the touchscreen or a mouse. A short tap (left mouse click) expands a menu or submenu. If a menu command has no submenu assigned, a short tap (left mouse click) opens a dialog, or directly activates the menu command. When a menu item is selected, the R&S ZNB displays the corresponding softtool.

Overview of menu functions

- The "File" menu provides functions to handle recall sets and trace files, to print or save diagram content, and to shut down the VNA application.
- The "Trace" menu provides all trace settings, the limit check settings, and the marker functions including marker search.
- The "Channel" menu provides all channel settings and activates, modifies or stores different channels.
- The "Display" menu provides all display settings and the functions for activating, modifying and arranging different diagrams.
- The "Tools" menu gives access to tools that extend the functionality of the analyzer firmware.

- The "System" menu provides functions that can be used to return to a defined instrument state, reverse operations, access service functions and define various system-related settings.
- The "Help" menu aids with the network analyzer and its operation.

You can configure the position of the menu bar (top/bottom) from the context menu of the softtool or hardkey panel. Furthermore, you can toggle its visibility using System – [Display] > "View Bar" > "Menu Bar".

3.3.2.6 Status bar

The status bar shows:

- The active channel
- The current channel's sweep averaging counter (e.g. "Ch<i>: Avg 9/10"), or "Ch<i>: Avg None" if averaging is disabled.
- The progress of the sweep
  The progress bar also shows when the R&S ZNB prepares a sweep with new channel settings.
  (See Section 4.1.4, "Sweep control", on page 82)
- The "EXT REF" symbol, if an external reference clock is used for synchronization.
  (See "Ext Frequency" on page 842)
- A symbol for redefined S-parameters, if the physical ports have been redefined.
  (See Section 5.17.8.2, "Define Physical Ports dialog", on page 864)
- The switch matrix status symbol, if a switch matrix is configured.
  (See Section 4.7.30, "External switch matrices", on page 271)
- The current date and time



The progress bar shows a moving color gradient if the current sweep is too fast to be monitored, e.g. because the number of sweep points is low.

Additional icons in the status bar indicate a non-standard hardware setup.

Table 3-4: Additional status bar information

Icon	Description
	External reference clock
	Redefined physical ports A double-click opens the Define Physical Ports dialog.
	Switch matrix A double-click opens the Switch Matrix RF Connections dialog.

You can configure the position of the status bar (top/bottom) from the context menu of the softtool or hardkey panel. Furthermore, you can toggle its visibility using System – [Display] > "View Bar" > "Status Bar".

3.3.3 Touchscreen gestures

Instruments of the 2nd generation (order nos 1334.3330.xx) are equipped with capacitive screens, which allow for multi-touch gestures.

A touchscreen allows you to interact with the software using various finger gestures on the screen. The basic gestures supported by the software and most applications are described here. Further actions using the same gestures may be possible.



Tapping

Touch the screen quickly, usually on a specific element.

You can tap most elements on the screen; in particular, any elements you can also click on with a mouse pointer.

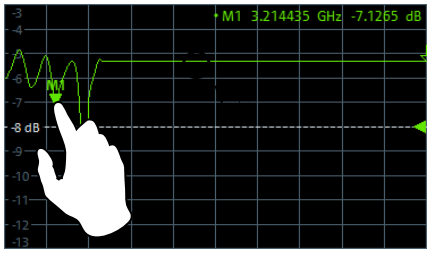
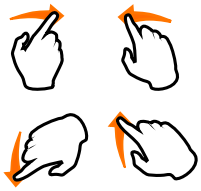


Figure 3-7: Tapping

Double-tapping

Tap the screen twice, in quick succession.

Double-tap a diagram to maximize it or to restore its original size.



Dragging

Move your finger from one position to another on the display, keeping your finger on the display the whole time.

By dragging your finger over a table or diagram you can pan the displayed area of the table or diagram to show results that were previously out of view.

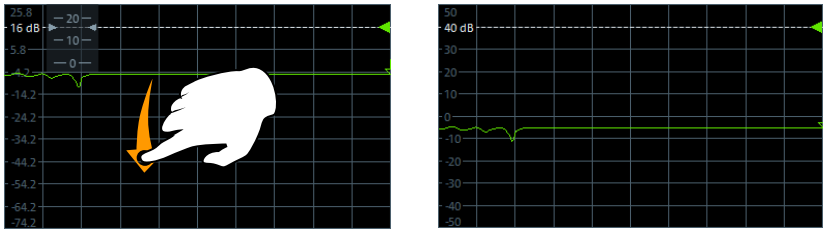
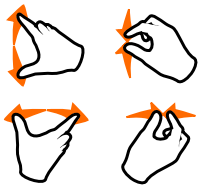


Figure 3-8: Dragging

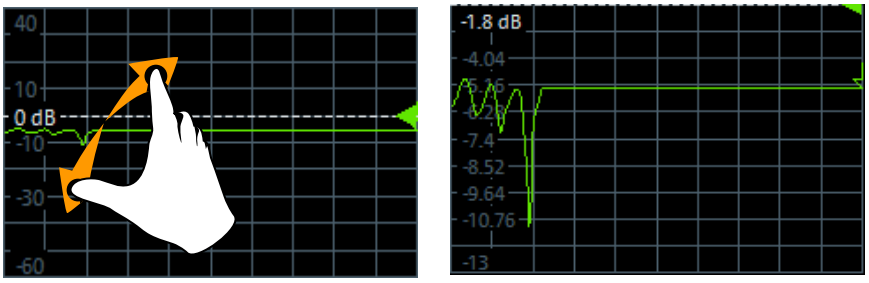


Spreading and pinching two fingers

Move two fingers apart on the display (spread) or move two fingers together on the display (pinch).

These gestures take effect for diagrams only. The effect depends on the current zoom mode (see Section 3.3.7, "Scaling diagrams", on page 65).

- In graphical zoom mode, when you spread two fingers in the display, you graphically zoom in vertically.



When you pinch two fingers in the display, you graphically zoom out vertically.

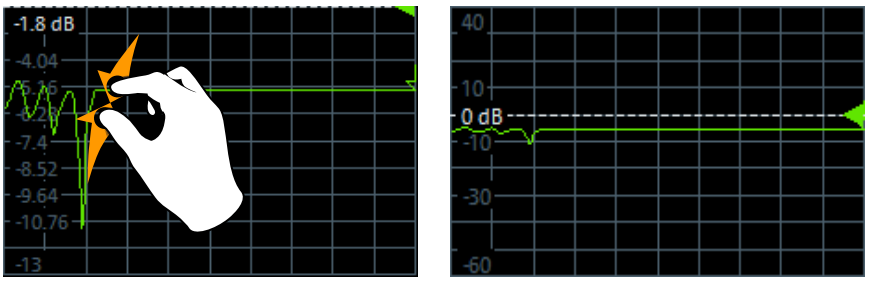


Figure 3-9: Pinching

While "Zoom Select" is active (toolbar icon or softtool button is toggled on), spreading and pinching is disabled. You can only select a rectangular area (using one finger) then.

- In stimulus zoom mode, spreading and pinging is disabled by default. Only while "Stim. Zoom Select" is active (toolbar icon or softtool button is toggled on), you can use spreading and pinching to modify the sweep area and/or the vertical scaling.

Mouse vs. touch actions

Any user interface elements that react to actions by a mouse pointer also react to finger gestures on the screen, and vice versa. The following touch actions correspond to mouse actions:

Table 3-5: Correlation of mouse and touch actions

Mouse operation	Touch operation
Click	Tap
Double-click	Double-tap
Click and hold	Touch and hold
Right-click	Touch, hold for 1 second and release

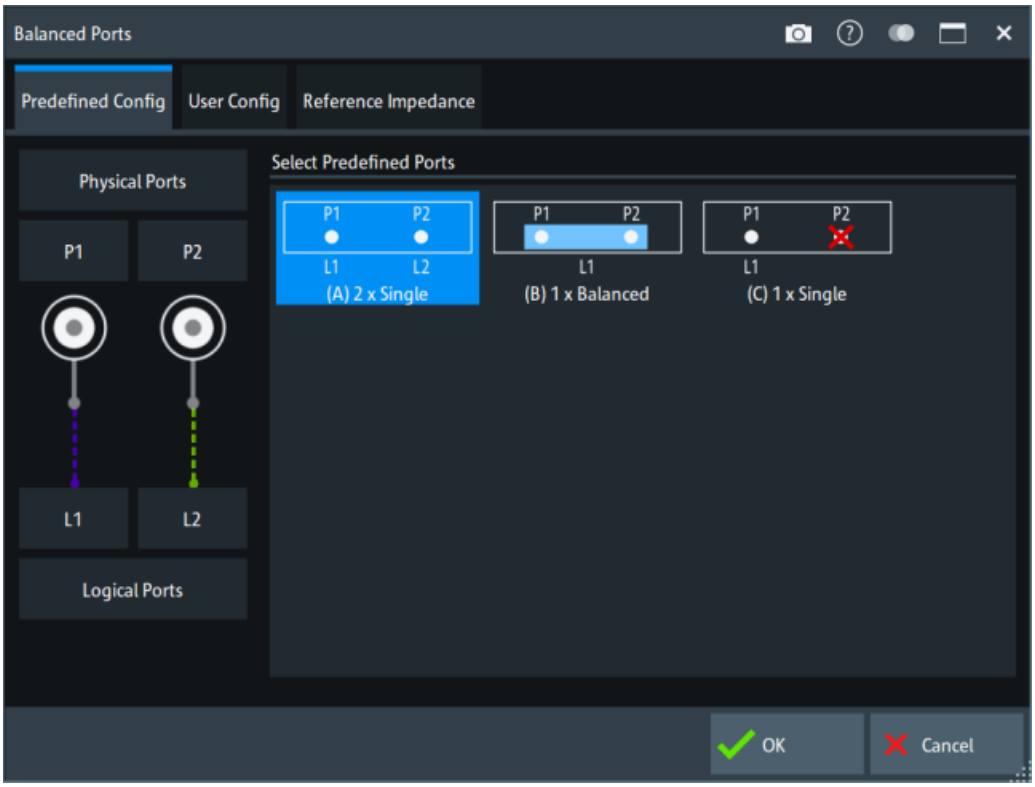
Mouse operation	Touch operation
Drag&drop (= click and hold, then drag and release)	Touch, then drag and release
n.a. (Change hardware settings)	Spread and pinch two fingers
Mouse wheel to scroll up or down	Swipe
Dragging scrollbars to scroll up or down, left or right	Swipe
In (graphical) zoom mode only: dragging the borders of the displayed rectangle to change its size	Touch, then drag and release

Example:

You can scroll through a long table in conventional mouse operation by clicking in the table's scrollbar repeatedly. In touchscreen operation, you can scroll through the table by dragging the table up and down with your finger.

3.3.4 Working with dialogs

Dialogs provide groups of related settings and allow you to make selections and enter data in an organized way. The settings are visualized, if possible. An example is shown below.



All dialogs are operated in a similar way.

- The title bar of each dialog contains some convenience functions:

– Use the "Opacity" function to let the display elements behind the dialog shine through.



Note: This is a global setting, i.e. it applies to all dialogs.

– The icon creates a screenshot of the dialog.
– The button opens the help system, which provides context-sensitive information on the dialog's settings.

- Drag and drop the lower right corner of the dialog to modify its size.
- Some dialogs are subdivided into tabs, containing groups of related settings. Activate a tab to access those settings.

See also Section 4.2.2.1, "Immediate vs. confirmed settings", on page 105.

3.3.5 Handling diagrams, traces, and markers

The analyzer displays measurement results as traces in rectangular diagrams. Markers are used to read specific numerical values and to search for points or regions on a trace. The following section presents some of the graphical tools the R&S ZNB provides for trace and marker handling.

For further reference

Refer to Section 4, "Concepts and features", on page 79 to learn more about traces, channels, and screen elements.

3.3.5.1 Adding traces and diagrams

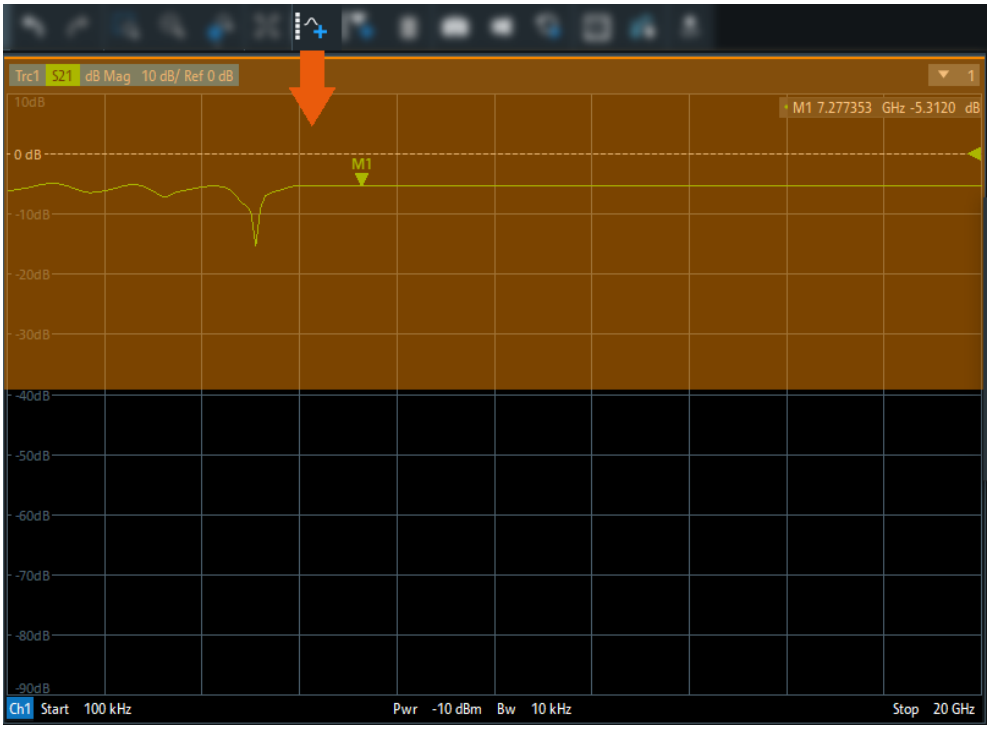
A new trace is required if you want to measure and display an additional quantity.

Typical scenario: The transmission coefficient S_{21} is measured as described in Section 3.4.1, "Transmission S-parameter measurement", on page 71. A trace is added to display the reflection coefficient S_{11} for comparison.

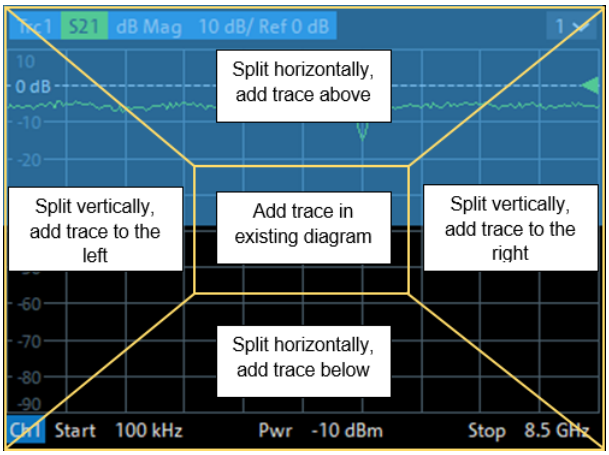
To create a trace:

1. Drag the icon from the toolbar into a diagram.

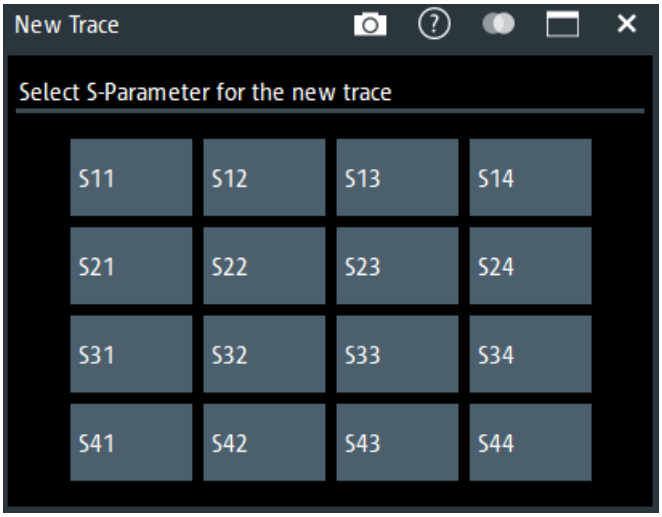
   The diagram changes its color scheme and contents as shown below.



2. Select the adequate drop position, depending on whether you want to display the new trace in the existing diagram, or whether you want to add a new diagram. The highlighted area indicates the target diagram.



3. In the dialog box that opens when you release the "New Trace" icon, select the S-parameter to be measured.
   For a four-port analyzer:



The R&S ZNB generates a new trace for the selected S-parameter.

Alternative control elements

You can also drag and drop a softkey representing a measured quantity to create a trace. Or simply select another softkey to change the measured quantity of the active trace.

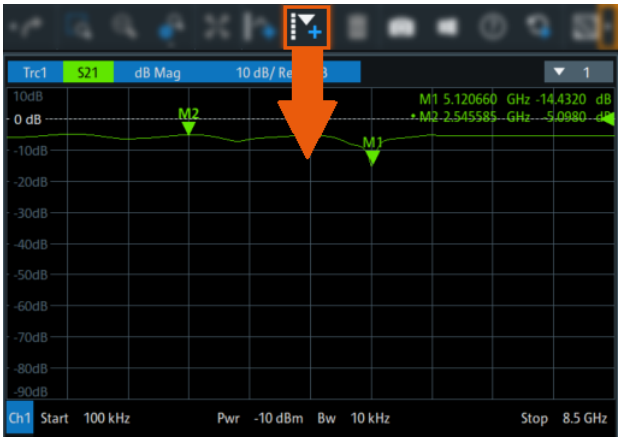
Select Trace – [Trace Config] to access more trace handling functions. Select System – [Display] to access more diagram handling functions.

3.3.5.2 Adding markers

A marker is needed, for instance, to read a particular numerical trace value.

To add a new marker:

- Drag and drop the icon from the toolbar to the desired position in the target diagram.



The marker appears on the target diagram's active trace. The marker info field displays the stimulus value (x-axis value) and response value (y-axis value) at the

marker position. The response value varies as the analyzer continues performing sweeps.



Active trace, alternative control elements

The trace line of the active trace in the upper part of the diagram is highlighted. If the diagram contains several traces, first activate the target trace, then add the marker.

The Trace – [Marker] softtool provides more functions for marker handling. In particular, any marker offered in the "Markers" tab can be positioned on the active trace using drag & drop.

3.3.5.3 Deleting display elements

Markers, traces, diagrams, and other display elements are most conveniently deleted using the "Delete" icon in the toolbar above the diagram area.

- To delete a single marker, drag it in vertical direction to release it from the trace and drop it onto the "Delete" icon.
  To delete a set of markers, drag and drop their marker info field onto the "Delete" icon.
  Deleting a marker and its info field also disables the associated marker function.
- To delete a trace, drag and drop its trace line onto the "Delete" icon. The active trace can also be deleted by tapping/clicking the "Delete" icon.
  Note however, that the last remaining trace cannot be deleted.
- To delete a diagram, drag and drop its dropdown handle onto the "Delete" icon.
  Note however, that the last remaining diagram cannot be deleted.
- To delete a channel, drag and drop all associated traces onto the "Delete" icon.
  Note however, that the last remaining channel cannot be deleted.
- To hide the limit lines and disable the limit check, drag and drop the PASS / FAIL message onto the "Delete" icon. The limit line itself is not deleted and can be reused.

The context menu of some display elements also provides the "Delete" function.

Undo function

If you happen to delete a display element unintentionally, you can restore it using the "Undo" toolbar icon.

3.3.5.4 Using drag and drop

You can drag and drop many of the R&S ZNB's control and display elements to change their size and position. The drag and drop functionality is often more convenient to use than the equivalent buttons of the softtool panels. The following table gives an overview.

Table 3-6: Drag and drop functionality for various screen elements

Screen element	Action	Drag and drop...
Diagram	Create	See Section 3.3.5.1, "Adding traces and diagrams", on page 57
	Resize	Separator between adjacent diagrams
	Delete	See Section 3.3.5.3, "Deleting display elements", on page 60
Trace	Create	See Section 3.3.5.1, "Adding traces and diagrams", on page 57
	Move vertically	Reference line marker (right diagram edge)
	Move into other or new diagram	Trace line
	Delete	See Section 3.3.5.3, "Deleting display elements", on page 60
	Reset / suspend graphic zoom	"Zoom" element in additional trace line --> "Delete" icon; see Section 3.3.7.1, "Using the graphical zoom", on page 65
Marker	Create	See Section 3.3.5.2, "Adding markers", on page 59
	Move horizontally	Marker symbol
	Delete	Marker or marker info field --> "Delete" icon; see Section 3.3.5.3, "Deleting display elements", on page 60
Marker info field	Add	See Section 3.3.5.2, "Adding markers", on page 59
	Move within diagram	Marker info field (move to one of several predefined positions)
	Delete	See Section 3.3.5.3, "Deleting display elements", on page 60

3.3.6 Entering data

The analyzer provides dialogs with various types of input fields where you can enter numeric values and character data. Data entry with a mouse and an external keyboard is a standard procedure known from other Windows® applications. However, there are various other ways to enter data.

3.3.6.1 Using front panel keys

You can use the keys in the data entry keypad to enter numbers, units, and characters.



To enter a numeric value:

1. Select a numeric data input field to activate it.
2. Press the data entry keys.
   - Use [0] to [9] to enter the corresponding numbers.
   - Use [.] to enter a decimal point.
   - Use [-] to change the sign of the value.
   - Use [G/n], [M/µ], [k/m], or [x1] to multiply the entered value with factors of $10^{(-)9}$, $10^{(-)6}$, $10^{(-)3}$ or 1 and to add the appropriate physical unit.

To enter a character string:

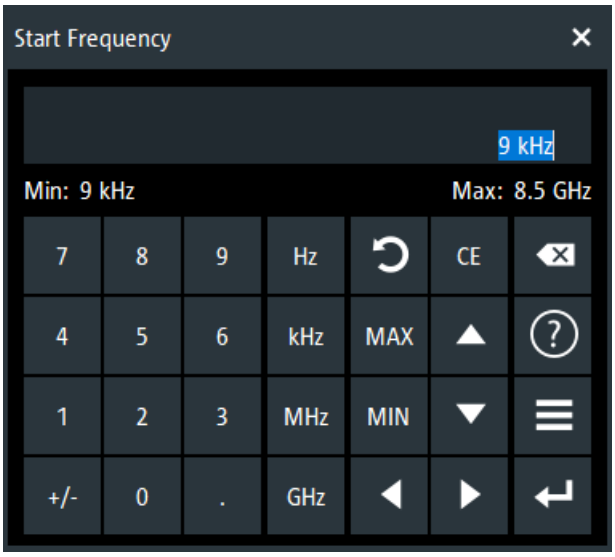
1. Tap a character data input field to activate it.
2. Press the data entry keys as if you were writing a short message on your mobile phone.
   - Use [0] to [9] to enter the corresponding numbers.
   - Use [.] and [-] to enter a dot or hyphen.
   - Use [G/n], [M/µ], or [k/m] to enter the letters G, M or K (case-insensitive).
   - Use the ⌫ (backspace) key to correct wrong entries, deleting the character to the left of the current cursor position.
   - Press [↵] (Enter) to complete an entry.
   - Press [Esc] to discard the entries made.
3. To enter letters other than G, n, M, µ, K or m you can also use one of the methods described in the following sections:
   - Section 3.3.6.3, "Using the analyzer's on-screen keyboard", on page 63
   - Section 3.3.6.4, "Using the Windows® on-screen keyboard", on page 64

3.3.6.2 Using the numeric editor

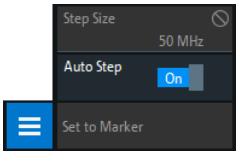
The "Numeric Editor" is a tool for convenient entry and modification of numeric values. It is available for all numeric input fields in the analyzer GUI.

Operation with touchscreen or mouse:

1. Double-tap (double-click) a numeric input field in a dialog or on a softtool to open the numeric editor.



2. Use the buttons in the numeric keypad to enter a value.
3. Add a unit (buttons"Hz", ..., "GHz")
4. Use the up/down buttons to increment/decrement the value by the current "Step Size".
5. Use the menu button to expand the dialog:

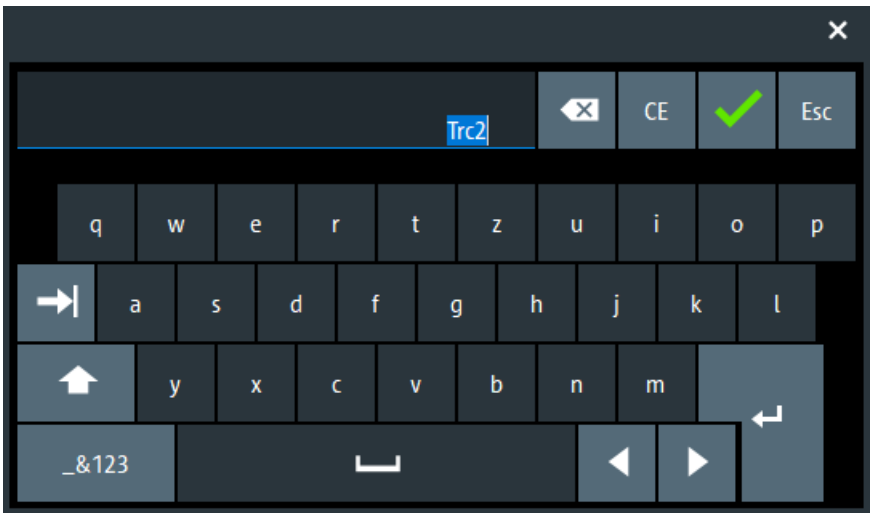


   This unhides the functions to set the "Step Size" and the "Set to Marker" function. The latter is enabled if the active trace has markers with a suitable stimulus or response unit.

6. After completing the input, select ↵ to apply the new value and close the editor.

3.3.6.3 Using the analyzer's on-screen keyboard

The on-screen "Keyboard" allows you to enter characters, in particular letters, without an external keyboard. It is available for all text input fields in the analyzer GUI.



1. Activate a character data input field in a softtool or a dialog.
2. Double-tap/click the input field to open the on-screen keyboard.
3. Select character buttons to compose the input string.
4. Select "Enter" to apply your selection and close the keyboard.

3.3.6.4 Using the Windows® on-screen keyboard

The Windows® on-screen keyboard allows you to enter characters, in particular letters, even if an input field cannot call up the analyzer's own on-screen keyboard. Examples are input fields in standard Windows ® dialogs.

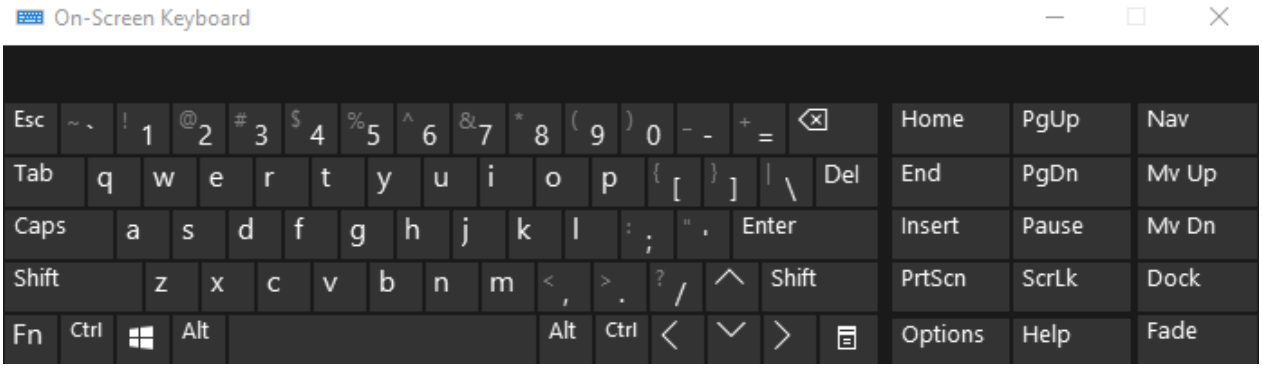


Figure 3-10: Windows on-screen keyboard

To call up the on-screen keyboard:

1. Open the System – [Applic] softtool.
2. Select "External Tools" > "Screen Keyboard".

3.3.7 Scaling diagrams

The analyzer provides various tools for customizing the diagrams and for setting the sweep range. Choose the method that is most convenient for you.

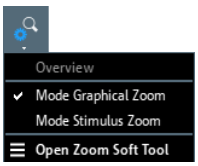
3.3.7.1 Using the graphical zoom

The graphical zoom function magnifies a rectangular portion of the diagram (zoom window) to fill the entire diagram area. The sweep points are not affected.

The graphical zoom function is only supported for cartesian trace formats. For (inverted) Smith and polar diagrams, it is not available.

To activate the graphical zoom:

- Select the "Zoom Config" icon in the toolbar above the diagram area and make sure that the graphical zoom mode is active:

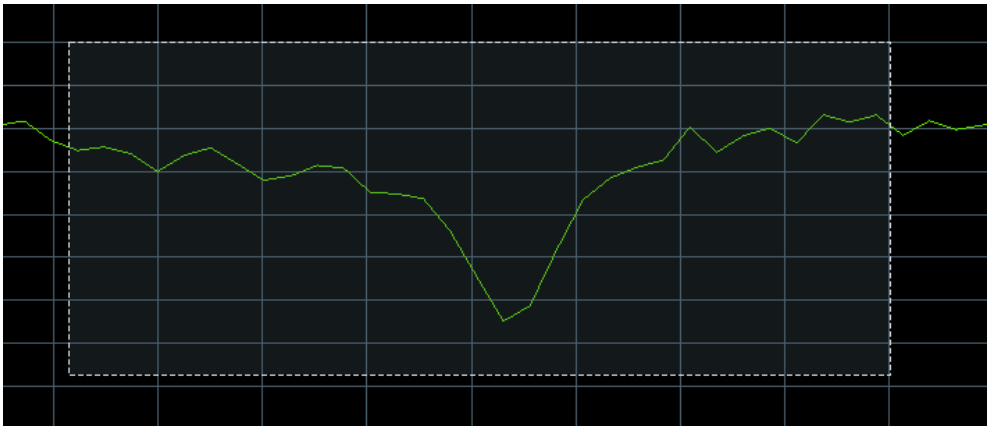


- Select the "Zoom Select" toggle button in the toolbar above the diagram area.

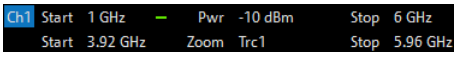


The icon changes its background color from black to blue.

- In the active diagram area, select an appropriate rectangular area.



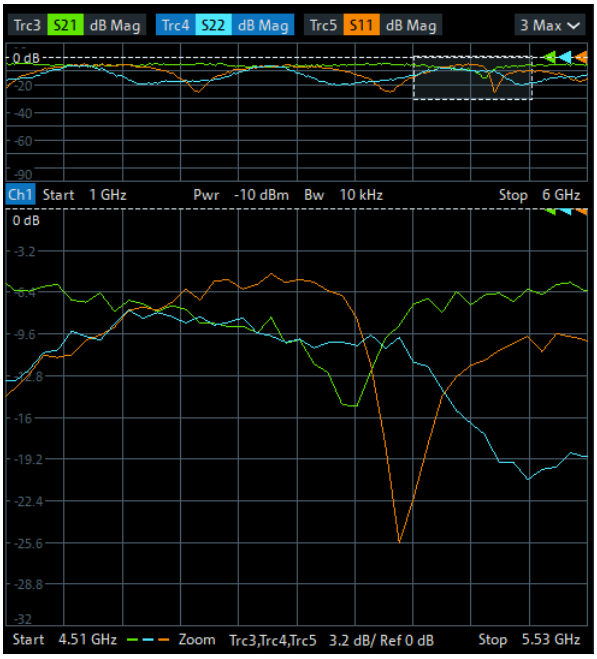
The zoomed view shows the selected rectangle, scaled in both horizontal and vertical direction. In general, the zoom window covers only a part of the sweep range; the horizontal distance between the sweep points increases. The reduced display range is indicated in an additional zoom line in the channel info area.



Use the "Zoom Reset" icon to restore the original diagram. Alternatively, you can drag and drop the "Zoom" label from the additional channel info line onto the toolbar button.

Alternative settings

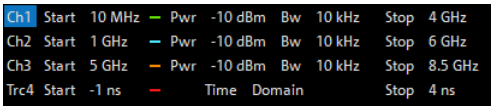
- The Trace – [Scale] > "Zoom" softtool tab allows you to define the displayed zoom range numerically.
  It can also be opened from the menu of the "Zoom Config" toolbar icon.
- If the active diagram is graphically zoomed, "Overview On" in the "Zoom" softtool tab toggles an overview. The upper part of the diagram then shows a small version of the unzoomed diagram. You can move the zoomed part of the trace by moving the rectangular area in the overview.



- To zoom the stimulus range (keeping the number of sweep points constant), use the "Zoom Config" toolbar icon and select "Mode Stimulus Zoom". Then use the "Zoom Select" icon to narrow the sweep range and adjust the vertical scaling.

3.3.7.2 Setting the sweep range

The sweep range for all related channels is displayed in the channel info area at the bottom of each diagram:



To change the sweep range of the active channel, use one of the following methods:

- Use the [Start], [Stop], [Center], and [Span] function keys from the Stimulus section.

- Double-tap (with a mouse: double-click) the "Start" or "Stop" label in the channel list.
- Tap and hold (with a mouse: right-click) the "Start" or "Stop" label in the channel list and select "Start Frequency", "Stop Frequency", "Center Frequency", or "Frequency Span" from the context menu.
- Select "Start Frequency", "Stop Frequency", "Center Frequency", "Span Frequency" from the "Channel" > "Stimulus" menu.
- Use the "Set by Marker" functions (Trace – [Marker] > "Set by Marker"; see Section 3.3.7.6, "Set by marker", on page 68).

3.3.7.3 Reference value and position

The analyzer provides three parameters for changing the scale of the vertical (response) axis:

- Changing the "Ref Value" or "Ref Pos" shifts the trace in a vertical direction and adjusts the labels of the vertical axis. "Ref Value" also works for radial diagrams.
- Changing the "Scale/Div" modifies the value of the vertical or radial diagram divisions and thus the entire range of response values displayed.

The "Scale/Div" and the "Ref Value" are indicated in the scale section of the trace info. In the example below, a "Scale/Div" of 10 dB and a "Ref Value" of 0 dB is used.



To change such a scale parameter, use one of the following methods:

- Open the Trace – [Scale] > "Scale Values" softtool tab and proceed from there.
- Tap and hold (with a mouse: right-click) the scale section in the trace info and select a setting from the context menu.
- Select a setting from the "Trace" > "Scale" menu.
- Use "Set by Marker" functions (Trace – [Marker] > "Set by Marker"; see Section 3.3.7.6, "Set by marker", on page 68).

3.3.7.4 Auto scale

The "Auto Scale" function adjusts the scale divisions and the reference value so that the entire trace fits into the diagram. To access "Auto Scale", use one of the following methods:

- Open the Trace – [Scale] > "Scale Values" softtool tab and select "Auto Scale Trace" or "Auto Scale Diagram".
- Tap and hold (with a mouse: right-click) the scale section in the trace info and select "Auto Scale Trace" from the context menu.
- Select "Auto Scale Trace" or "Auto Scale Diagram" from the "Trace" > "Scale" menu.

3.3.7.5 Circular diagrams

The radial scale of a circular diagram ("Polar", "Smith" or "Inverted Smith") can be changed with a single linear parameter, the "Ref Value". The reference value defines the radius of the outer circumference.

- Increasing the "Ref Value" scales down the polar diagram.
- Decreasing the "Ref Value" magnifies the polar diagram.

The "Ref Value" is indicated in the scale section of the trace info.

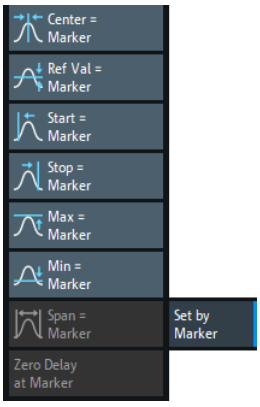


To change the "Ref Value" setting, use one of the following methods:

- Locate it on the Trace – [Scale] > "Scale Values" softtool tab.
- Tap and hold (with a mouse: right-click) the scale section in the trace info and select the parameter from the context menu.
- Select the parameter from the "Trace" > "Scale" menu.
- Use the "Set by Marker" functions; see Section 3.3.7.6, "Set by marker", on page 68.

3.3.7.6 Set by marker

The "Set by Marker" functions are a convenient tool for scaling (in particular: magnifying) diagrams without entering explicit numeric values. You simply place a marker to a trace point and use the marker values to change the sweep range or move the trace relative to the vertical axis. The touchscreen or a mouse makes it easier to activate (touch/click) or move (drag and drop) markers.



To set the sweep range using markers, use one of the following methods.

Set "Start" and "Stop" values in the diagram:

1. Create two normal markers, e.g. the markers "Mkr 1" (default label "M1") and "Mkr 2" (default label "M2").
   See Section 3.3.5.2, "Adding markers", on page 59.
2. Place "M1" to the start value of the desired sweep range and tap Trace – [Marker] > "Set by Marker" > "Start = Marker".

3. Place "M2" to the stop value of the desired sweep range and tap Trace – [Marker] > "Set by Marker" > "Stop = Marker".

Use a definite "Span":

1. Create a marker.
2. Enable "Delta Mode" for this marker.
   The analyzer automatically creates an additional reference marker.
3. Place the reference marker to the start value of the desired sweep range.
4. Set the value of the delta marker to the desired sweep range and tap Trace – [Marker] > "Set by Marker" > "Span = Marker".

To move the trace in a vertical direction, proceed as follows:

1. Create a normal marker, e.g. the marker "Mkr 1" (default label "M1").
2. Place "M1" to a particular trace point, e.g. use the "Marker Search" functions to locate a maximum or minimum on the trace.
3. Select Trace – [Marker] > "Set by Marker" > "Max = Marker" to move the trace towards the upper diagram edge, leaving the values of the vertical divisions ("Scale/Div") and the overall vertical scale unchanged. Analogously, select "Min = Marker" to move the trace towards the lower diagram edge, or select "Ref Val = Marker" to move the trace towards the "Ref Value".



You can also use marker values in the "Numeric Editor"; see Section 3.3.6.2, "Using the numeric editor", on page 63.

3.3.7.7 Enlarging a diagram

The analyzer provides different tools for customizing the contents and size of the diagrams:

- Select ⛶ from the toolbar to maximize the active diagram. Or, equivalently, select System – [Display] > "Diagram" > "Maximize Diagram".
  If enabled the active diagram is always maximized.
- Double-tap/click a diagram to maximize it and make all its traces except the active one invisible.
- The "Menu Bar", the "Status Bar", the "Hard Key Panel", and the "Title Bar" can be hidden to gain space for the diagrams (System – [Display] > "View Bar").
- The System – [Display] > "Config" softtool tab defines optional display elements for the interior of the diagrams.

Use the context menu of the diagram, the System – [Display] key or the "Display" menu to access the display settings.

3.4 Performing measurements

This section takes you through a sample session with a R&S ZNB network analyzer and describes basic operation tasks.

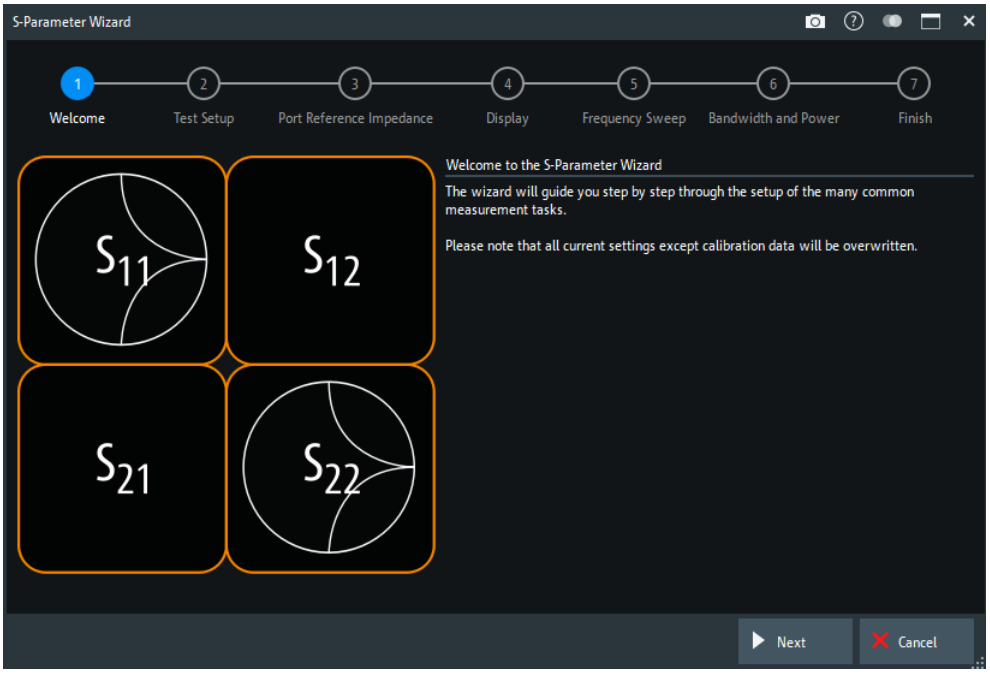
Prerequisite

The instrument is set up, connected to the mains system, and started up as described in Section 3.1, "Preparing for use", on page 21.

Use the "S-Parameter Wizard" accessible via

Trace – [Meas] > "Measurement Menu: S-Parameters" > "S-Params" > "S-Param Wizard"

to measure S-parameters in a straightforward way. The wizard provides a series of dialogs where you can select the test setup, screen configuration and measurement parameters, configure the essential channel settings and perform a guided calibration.



Measurement stages in the wizard

The individual dialogs of the "S-Parameter Wizard" correspond to the typical stages of any measurement:

1. Prepare the measurement.
2. Select the test setup.
3. Define port impedances.
4. Select the measurement parameters and the diagrams.

5. Define the sweep range.
6. Adjust the source and receiver settings (source power, measurement bandwidth).
7. Perform a calibration (optional).

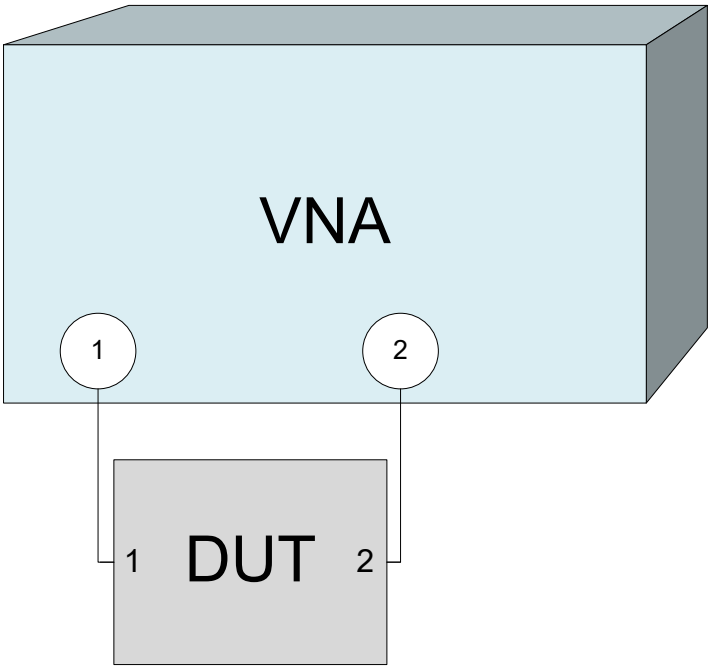
3.4.1 Transmission S-parameter measurement

In a transmission measurement, the analyzer transmits a stimulus signal to the input port of the device under test (DUT) and measures the transmitted wave at the DUT's output port. The trace settings allow you to select the measured quantities and display formats, depending on what you want to learn from the data. A minimum of two analyzer test ports are required for transmission measurements.

In the following example, the analyzer is set up for a two-port transmission measurement. A frequency sweep range is selected, the instrument is calibrated and the measurement result is analyzed using various display formats.

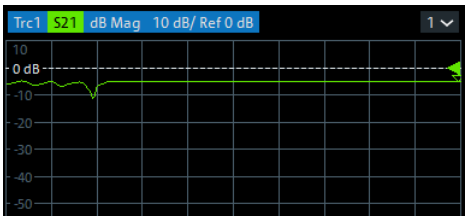
3.4.1.1 Connecting the instrument for transmission measurements

To prepare a transmission measurement, you have to connect your DUT (which for simplicity we assume to have appropriate connectors) in-between a pair of analyzer test ports. It is recommended that you preset the R&S ZNB to start from a well-defined instrument state.



1. Connect the DUT between test ports 1 and 2 of the network analyzer as shown above.
2. Use the [Preset] key to restore a well-defined instrument state.

The analyzer is now set to its default state, measuring the transmission S-parameter S_{21}.



Drag&drop the toolbar icon onto the diagram area to create additional traces (and diagrams).

3.4.1.2 Selecting the sweep range and other parameters

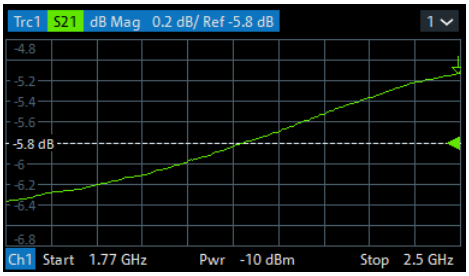
After a system preset the display shows a diagram with a dB magnitude scale, and the S-parameter S_{21} is selected as a measured quantity. This S-parameter is the forward transmission coefficient of the DUT. It is defined as the ratio of the transmitted wave at the DUT's output port (port no. 2) to the incident wave at the DUT's input port (port no. 1).

The R&S ZNB automatically adjusts its internal source and receiver to the selected measured quantities: For an S_{21} measurement, a stimulus signal (termed a_1) is transmitted at the analyzer port no. 1; the transmitted wave (termed b_2) is measured at port 2. The stimulus signal from the analyzer port no. 2 is not needed except for some calibration types.

By default the sweep range is set to the frequency range of the analyzer, which can be unsuitable for your DUT. The following procedure shows you how to configure a smaller sweep range.

1. Select Stimulus – [Start] and set the "Start Frequency" to the lowest frequency you want to measure (e.g. 1.77 GHz).
   For convenient numeric entry, double-tap/click the input field to open the "Numeric Editor" (see Section 3.3.6.2, "Using the numeric editor", on page 63).

   Tip: If you use the data entry keys at the front panel for data entry, type [1][.][7][7] and terminate the entry with the [G/n] key.
   Refer to Section 3.3.6, "Entering data", on page 61 to learn more about entering numeric values and characters.
2. In the "Stop Frequency" input field, enter the highest frequency you want to measure (e.g. 2.5 GHz).
3. Select Trace – [Scale] > "Scale Values" and activate the "Auto Scale Trace" function. The analyzer adjusts the scale of the diagram to fit in the entire S_{21} trace, leaving an appropriate display margin.



Tip: Refer to Section 3.3.7, "Scaling diagrams", on page 65 to learn more about the different methods and tools for diagram scaling.

By default, data acquisition at port 2 starts immediately. For DUTs with large group delay, and if the sweep is optimized for speed, the data acquisition window can partly overlap with the signal from the previous sweep point. In this case, it is recommended to configure an additional Meas Delay: Start with a "Meas Delay" of 0, and increase it until the S_{21} trace becomes stable.

3.4.1.3 Calibrating the instrument

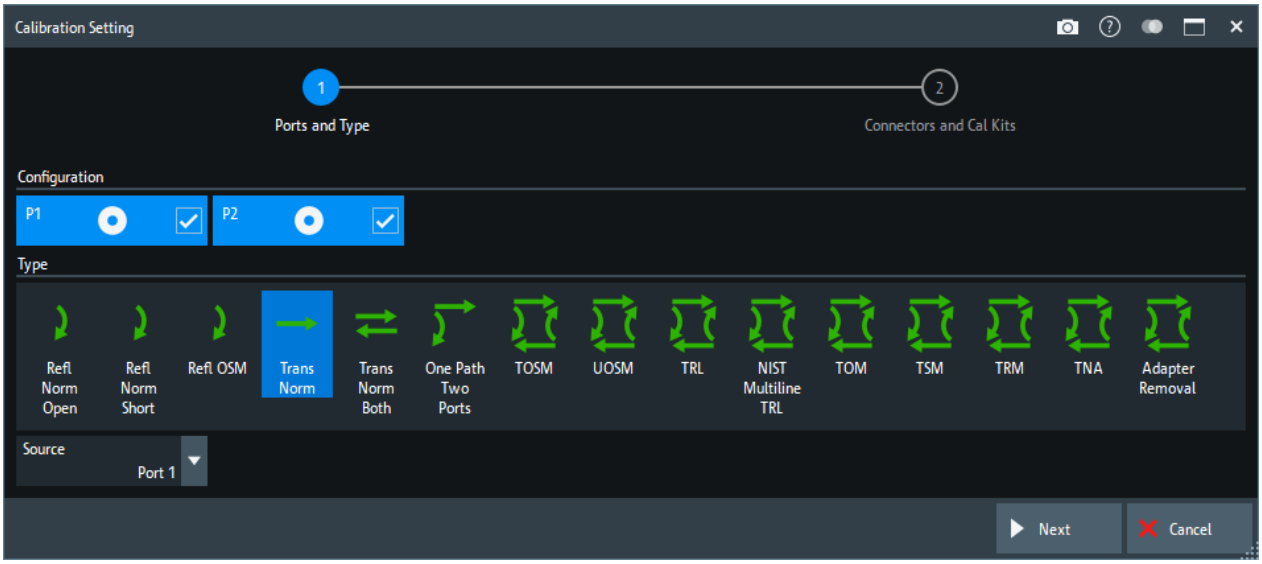
Calibration (system error correction) is the process of eliminating systematic, reproducible errors from the measurement results. E.g., in the current test setup, the connecting cables between the analyzer ports and the DUT introduce an attenuation and a phase shift of the waves. Both effects impair the accuracy of the S-parameter measurement.

The analyzer provides a wide range of sophisticated calibration methods for all types of measurements. The calibration method to select depends on the expected system errors, the accuracy requirements of the measurement, on the test setup and on the types of calibration standards available.

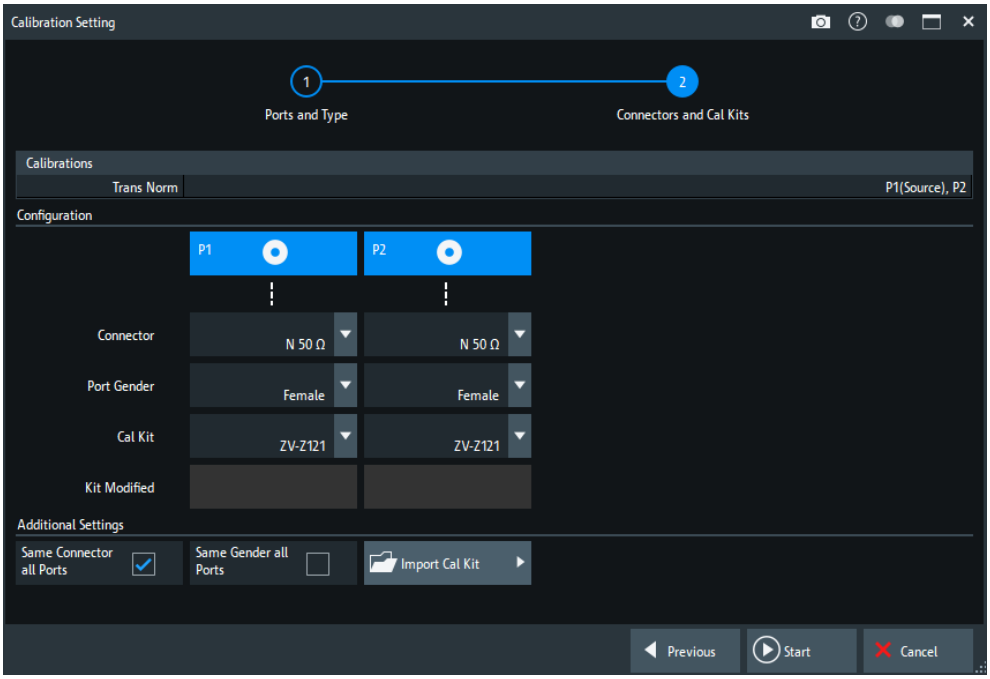
The following example requires a calibration kit with a male Through standard with known transmission characteristics for the related test port connector type and gender. With a single Through, it is possible to perform a transmission normalization, compensating for a frequency-dependent attenuation and phase shift in the signal paths.

Due to the R&S ZNB's calibration wizard, calibration is a straightforward, guided process.

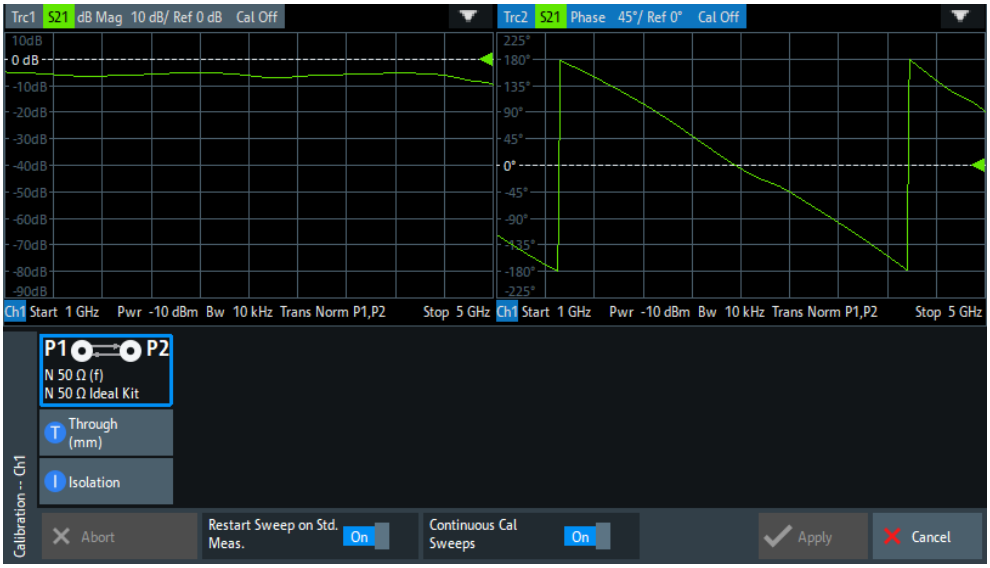
1. Replace the DUT by the Through standard of your calibration kit. Make sure to disconnect all calibration units.
2. Select Channel – [Cal] > "Start (Manual)" to open the "Calibration Setting" wizard.
3. Pick port 1 (P1) and port 2 (P2), choose the calibration type "Trans Norm", and select port 1 as source port.



4. Select "Next" to proceed to the next page of the "Calibration Setting" wizard.
5. Select the test port connector type and gender (here: N 50 Ω, female, corresponding to a male Through standard), and the calibration kit (here: R&S ZV-Z121).



6. Tap"Start".
7. The calibration dock widget indicates the standard measurements that make up a "Trans Norm" calibration.
   Select "Through (mm)" to initiate the measurement of the connected Through standard. Measuring the isolation between ports 1 and 2 is optional. Skip it for now.



The analyzer performs a calibration sweep for the measured quantity S_{21}. The magnitude and phase of the result is displayed in two diagrams, together with the expected typical result for a Through standard. The similarity of real and expected traces indicates that the Through standard has been properly connected. After the R&S ZNB has completed the calibration sweep and calculated the correction data, the "Apply" button is enabled.

8. Select "Apply" to close the wizard.

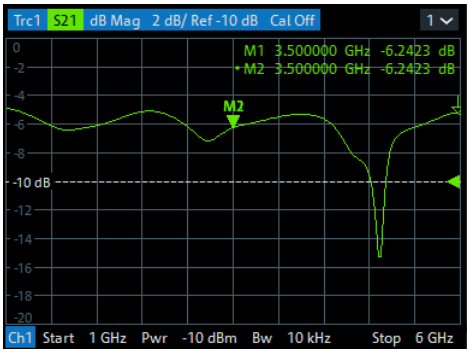
   The system error correction is calculated and applied to the current channel. A "Cal" label appears in the trace list.

To proceed with the measurement, remove the Through standard and connect the DUT again.

3.4.1.4 Evaluating data

The analyzer provides various tools to optimize the display and analyze the measurement data. For instance, you can use markers to determine maxima and minima on the trace, and change the display format to obtain information about the group delay of the transmitted wave.

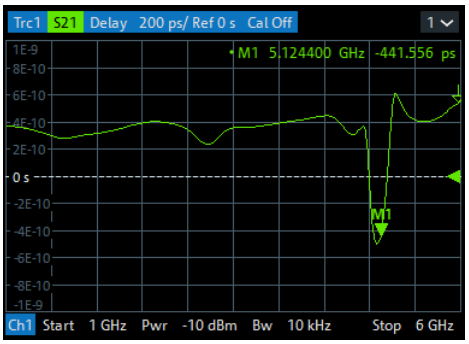
1. Select Trace – [Marker] > "Markers" > "Mkr 1".
   This places marker "M1" to its default position (center of the sweep range). A marker symbol (triangle) appears on the trace, a marker info field in the upper right corner of the diagram. The marker info field displays the stimulus value (frequency) and response value (magnitude of the transmission coefficient converted to a dB value) at the marker position.



2. Select Trace – [Marker], activate the "Marker Search" softtool tab and activate "Min" search.
   The marker jumps to the absolute minimum of the curve in the entire sweep range. The marker info field shows the coordinates of the new marker position.

3. Select Trace – [Format] and choose the "Delay" of the transmission coefficient as the displayed quantity.

   The group delay represents the propagation time of the wave through the DUT; it is displayed in a Cartesian diagram. The marker info field shows the frequency and group delay at the marker position.



Refer to Section 4.2.3, "Trace formats", on page 108 to learn more about the diagram properties.

3.4.1.5 Saving and printing data

The analyzer provides standard functions for saving measurement settings and for saving or printing the results. You can use these functions as if you were working on a standard PC. Moreover you can export your trace data to an ASCII file and reuse it in a later session or in an external application.

Data transfer is made easier if external accessories are connected to the analyzer or if the instrument is integrated into a LAN. Refer to Section 3.1.11, "Connecting external accessories", on page 29, and Section 3.1.12, "Remote operation in a LAN", on page 31 to obtain information about the necessary steps.

1. Activate the System – [File] > "Trace Data" softtool tab.
2. In the "Trace Data" softtool tab, select "Export" – "ASCII" to open the "Export Data - ASCII Files" dialog.
3. In the "Export Data - ASCII Files" dialog:
   a) Select a file location ("Look in:").
   b) Enter a filename ("File name:").
   c) Select "Save".

   The analyzer writes the data of the active trace to an ASCII file and closes the dialog.
4. Activate the "Print" softtool tab (System – [File] > "Print").
5. In the "Print" softtool tab, select "Print" to print the diagram area or "To Clipboard" to copy it to the Windows clipboard.
6. Select System – [File] > "Recall Sets" > "Save" to open the "Save" dialog for recall sets.
7. In the "Save" dialog:
   a) Select a file location ("Look in:").
   b) Enter a name for the recall set file ("File name:").
   c) Select "Save".

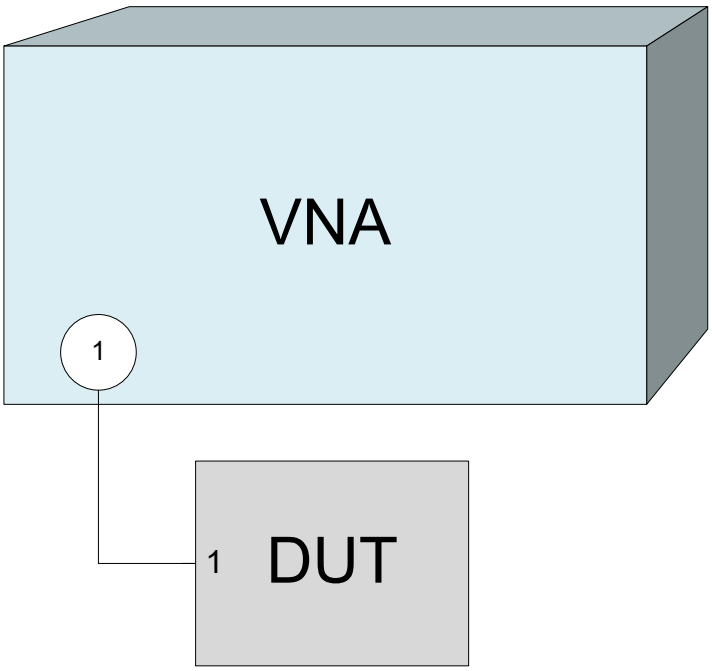
   The analyzer saves the active recall set, containing channel, stimulus and trace settings, to a `znx` file. This recall set can be restored in a later session.

3.4.2 Reflection S-parameter measurement

In a reflection measurement, the analyzer transmits a stimulus signal to the input port of the device under test (DUT) and measures the reflected wave. Different trace formats allow you to express and display the results, depending on what you want to learn from the data. Only one analyzer test port is required for reflection measurements.

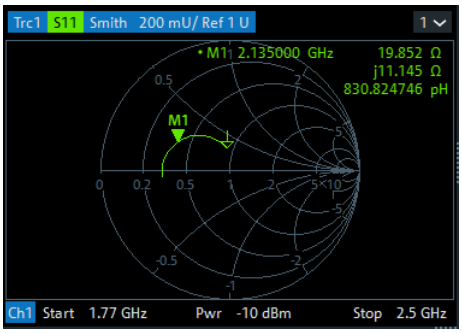
In principle, a reflection measurement involves the same steps as a transmission measurement. Note the following differences:

- The basic test setup for reflection measurements involves a single DUT and analyzer port. For instance, you can connect the input of your DUT to port 1 of the analyzer as shown below.



You can also use the basic transmission test setup, e.g. if you want to measure reflection and transmission parameters in parallel.

- The analyzer provides special calibration types for reflection measurements (Refl. Norm., OSM, ...). Use the calibration wizard and select an appropriate type.
  A full n-port calibration (TOSM, UOSM, TNA ...) corrects the system errors for all transmission and reflection S-parameters.
- Some of the trace formats are particularly suited for reflection measurements. For instance, you can display the measured reflection coefficient S_{11} in a Smith chart to obtain the complex input impedance at DUT port 1.



4 Concepts and features

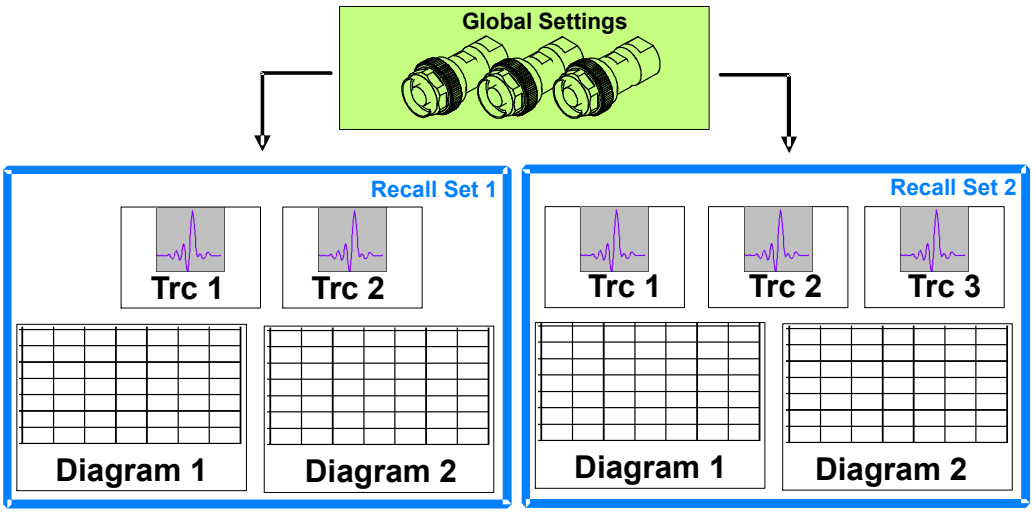
The following chapter provides an overview of the analyzer's capabilities and their use. It contains a description of the basic concepts that the analyzer uses to organize, process and display measurement data. Also included are descriptions of the screen contents, possible measured quantities, calibration methods and typical test setups.

For a systematic explanation of all softtools, functions and parameters refer to Section 5, "GUI reference", on page 289.

4.1 Basic concepts

The analyzer provides various functions to perform a particular measurement and to customize and optimize the evaluation of results. To ensure that the instrument resources are easily accessible and that user-defined configurations can be conveniently implemented, stored and reused, the instrument uses a hierarchy of structures:

- Global resources can be used for all measurements, irrespective of the current measurement session.
- A recall set comprises a set of diagrams together with the underlying system, channel, trace and display settings. It can be saved to a recall set file and later recalled.
- The diagrams show traces which are assigned to channels. See Section 4.1.3, "Traces, channels and diagrams", on page 80.



4.1.1 Global (persistent) settings

The analyzer manages global settings that apply to all measurements, irrespective of the current measurement setup. The following list contains examples of global settings:

- Calibration kits
- Connector types

- Calibration pool: system error correction and power correction data
- Directories for trace data, limit lines, calibration data etc.
- Color schemes and printer settings
- System configurations, to be accessed via System – [Setup].
- External power meter, generator and switch matrix configurations

Global settings are not part of a recall set nor are they affected by a [Preset] of the analyzer. Many of them can be "Reset" in the "System Config" dialog.

To reset all global settings in one go, exit the firmware and run the "Reset Factory State" script (Windows Start menu > "R&S ZNB Vector Network Analyzer" program group).

Some settings are session-specific, i.e. they are initialized to default when a new measurement session is started (session settings).

4.1.2 Recall sets

A recall set comprises a set of diagrams together with the underlying system, channel, trace and display settings. The R&S ZNB can handle multiple recall sets in parallel, each of them displayed in a separate tab.



A recall set can be saved to a recall set file (`*.znx|*.znxml`) and reopened at a later point in time or at another instrument. Use the "Recall Sets" tab of the System – [File] softtool to organize your recall sets.

A recall set only contains setup instructions, i.e. information on how to measure, how to process the measurement results, and how to display the processed results. It does **not** contain any trace or result data.

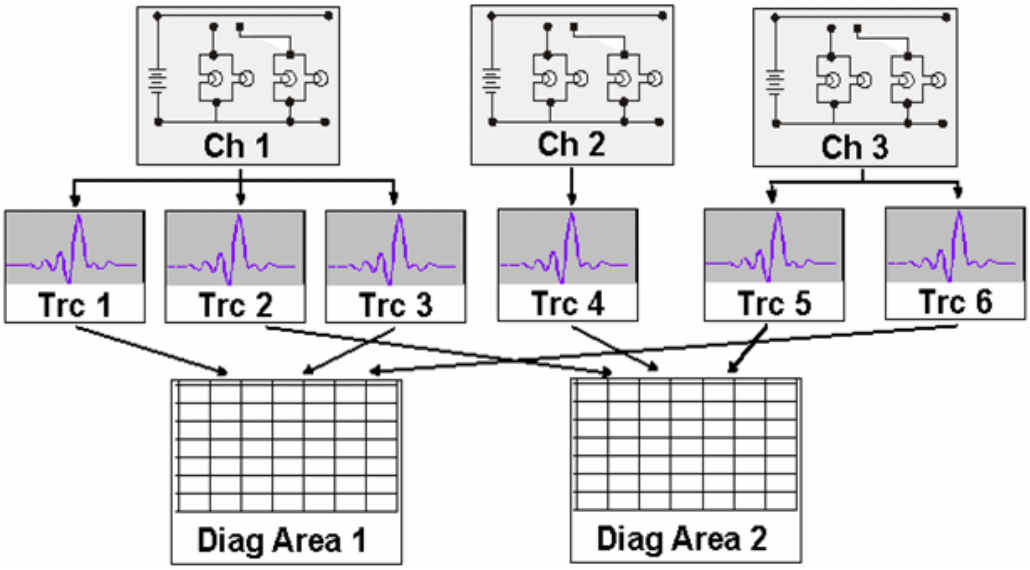
4.1.3 Traces, channels and diagrams

The analyzer arranges, displays or stores the measured data in traces which are assigned to channels and displayed in diagrams. To understand the functions of the instrument and quickly find the appropriate settings, it is important to understand the exact meaning of the three terms.

- A trace is a set of data points that can be displayed together in a diagram. The trace settings specify the mathematical operations used to obtain traces from the measured or stored data and to display them.
- A channel contains hardware-related settings which specify how the network analyzer collects data.
- A diagram is a rectangular portion of the screen which is used to display traces. Diagrams belonging to the same recall set are arranged in a common tab. The set-

tings for diagrams are described in Section 4.2.1, "Display elements of a diagram", on page 93.

A diagram can contain a practically unlimited number of traces, assigned to different channels. Diagrams and channels are independent from each other.



4.1.3.1 Trace settings

The trace settings specify the mathematical operations used to obtain traces from the measured or stored data. They can be divided into several main groups:

- Selection of the measured quantity (S-parameters, wave quantities, ratios, impedances,...)
- Conversion into the appropriate display format and selection of the diagram type
- Scaling of the diagram and selection of the traces associated to the same channel
- Readout and search of particular values on the trace by means of markers
- Limit check

The trace settings can be accessed via the keys in the Trace section of the (virtual) hardkey panel. They complement the Channel settings accessible via the Stimulus and Channel sections.

Each trace is assigned to a channel. The channel settings apply to all traces of the channel.

4.1.3.2 Channel settings

A channel contains hardware-related settings which specify how the network analyzer collects data. The channel settings can be divided into three main groups:

- Description of the test setup (power of the internal source, IF filter bandwidth, port configuration, receiver step attenuators, ...)
- Control of the measurement process (sweep, trigger, averaging, ...)

- Correction data (calibration, offset, ...)

The channel settings can be accessed via the Stimulus and Channel sections of the (virtual) hardkey panel.

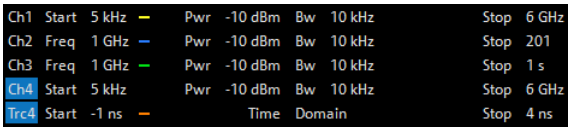
4.1.3.3 Active and inactive traces and channels

A window can display several diagrams simultaneously, each with a variable number of traces. One of these traces is active at each time. The **active trace** is highlighted in the trace list on top of the active diagram (Trc4 in the figure below):



When a trace is selected in the diagram area, it becomes the **active trace**. If a previously inactive area is selected as the active area, the trace that was active last time when the area was active again becomes the active trace.

The **active channel** is the channel which belongs to the active trace. The channels of all traces in a diagram are listed at the bottom of the diagram, together with the "Stimulus" values and the display colors of all traces. The active channel is highlighted.



Channels with no traces are not indicated in the diagrams but can be accessed via the "Channel Manager" dialog.

In manual control, there is always exactly one active trace, irrespective of the number of channels and traces defined. In remote control, each channel contains an active trace.

See also Section 6.3, "Basic remote control concepts", on page 882.

4.1.4 Sweep control

A sweep is a series of consecutive measurements taken over a specified sequence of stimulus values. It represents the basic measurement cycle of the analyzer.

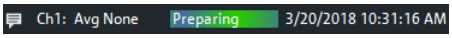
The analyzer can perform:

- Sweeps at constant power but variable frequency (frequency sweeps)
- Sweeps at constant frequency but variable power (power sweeps)
- Sweeps at constant power and frequency that are repeated in time (Time/CW Mode sweeps)

The sweeps are further specified by the number of measurement points and the total measurement time.

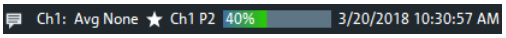
By default, sweeps are repeated continuously. Alternatively, a measurement can also consist of a single sweep or of a specified number of sweeps.

After changing the channel settings or selecting another measured quantity, the analyzer needs some time to initialize the new sweep. This preparation period increases with the number of points and the number of partial measurements involved. It is indicated in the status bar:



All analyzer settings can still be changed during sweep initialization. If necessary, the analyzer terminates the current initialization and starts a new preparation period.

During the first sweep after a change of the channel settings, the analyzer displays an asterisk symbol in the status bar.



4.1.4.1 Partial measurements and driving mode

Depending on the measurement task and the measured quantities, the measurement at each sweep point can consist of one or several "partial measurements" with definite hardware settings.

- If a single S-parameter is measured (e.g. the reflection coefficient S_{11}), the analyzer can operate at fixed hardware settings. In particular, a fixed source port and receive port are used. Each sweep point requires a single partial measurement. See also Section 4.3.1, "S-parameters", on page 117.
- For a complete two-port S-parameter measurement (e.g. S_{11}, S_{21}, S_{12}, S_{22}) the analyzer needs to interchange the roles of the source and receive ports. Each sweep point requires two partial measurements.

To improve the accuracy, it is possible to insert a delay time before each partial measurement.

In the **"Alternated"** driving mode (default), the R&S ZNB performs a partial measurement at all sweep points (partial sweep) before the hardware settings are changed. The next partial measurement is carried out in an additional sweep. In the **"Chopped"** driving mode, the order is reversed, i.e. the measurement proceeds sweep point per sweep point, with varying hardware settings.

See Channel – [Channel Config] > "Mode" > "Driving Mode".

Advantages of alternated and chopped driving mode

If the settling time between sweep points is smaller than the settling time between hardware settings (which is generally true), then the "Alternated" mode is faster than the "Chopped" mode. On the other hand, the "Chopped" mode builds up traces sweep point per sweep point, whereas in the "Alternated" mode you have to wait for the last partial sweep.

Use the "Alternated" mode to increase the accuracy of measurements on DUTs with long level settling times (e.g. quartzes, SAW filters). To measure DUTs with short settling times and obtain a trace from the beginning of the sweep, use "Chopped" mode. In "Auto" mode, the analyzer optimizes the display update: Fast sweeps are performed in "Alternated" mode, slower sweeps in "Chopped" mode.

As an alternative to activating the "Alternated" mode, it is possible to insert a measurement delay before each partial measurement and thus improve the accuracy.

See Channel – [Sweep] > "Sweep Params" > "Meas Delay".

However, the delay slows down the measurement.

Relation to trigger settings

In triggered measurements, "Alternated" has no effect if the triggered measurement sequence is identical to a single sweep point. The following table shows how the analyzer performs a sweep comprising *m* sweep points, assuming that each of them requires *n* partial measurements.

Triggered meas. sequence	"Alternated" mode	"Chopped" mode
Sweep	Trigger event starts *n* partial sweeps over all sweep points.	Trigger event starts *m* complete measurements at consecutive sweep points.
Sweep segment	Trigger event starts *n* partial sweeps over the next segment.	Trigger event starts complete measurements at all consecutive sweep points in the segment.
Point	All partial measurements of each sweep point are carried out one after another.	All partial measurements of each sweep point are carried out one after another.
Partial measurement	Each partial measurement is carried out for all sweep points.	All partial measurements of each sweep point are carried out one after another

Block Alternated driving mode

The "Block Alternated" driving mode allows you to split the sweep range into several (~ equally sized) blocks of sweep points. The measurement proceeds block per block, using the "Alternated" driving mode for each block.

If two or more partial measurements are required, you can minimize the overall sweep duration by selecting a suitable number of blocks.

4.1.4.2 Pipelining of measurement data

In the legacy sweep implementation, the next sweep begins when all trace data from the previous sweep has been processed.

In alternated driving mode, trace data processing is triggered when the first data of the last partial measurement arrives.

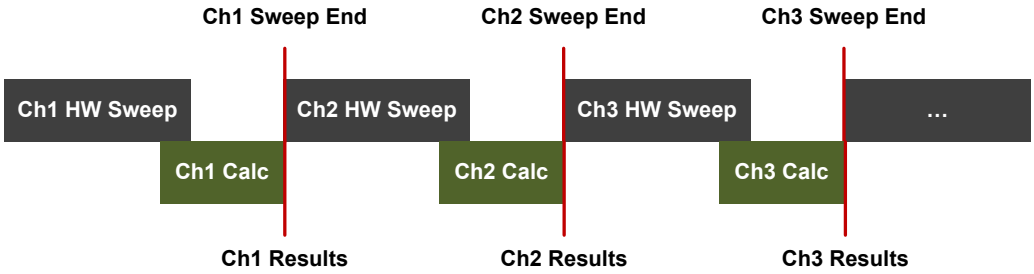


Figure 4-1: Legacy sweep implementation (no pipelining)

With pipelining, the next sweep can already start when the raw measurement data of the previous sweep has been distributed completely. The calculation of the previous sweep is processed in parallel to the next one, which effectively eliminates hardware idle time during sweeps.

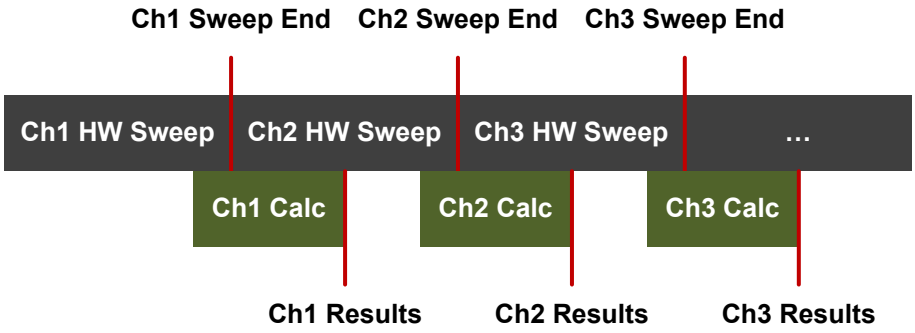


Figure 4-2: Pipelining

Pipelining is also available in chopped driving mode. However, chopped mode is typically much slower than alternated mode and the speed improvement due to pipelining is comparatively small.

In the current implementation ("setup-based pipelining"), calculations are synchronized to the end of the sweep sequence: when all channels of the active setup are swept, the firmware waits until all data has been calculated before starting the next sweep sequence.

With active pipelining, “the measurement result” (e.g. a pass/fail) will be available when the next measurement is already running. Depending on the measurement scenario, also a pipelined DUT handling can be necessary. For example, a device handler has to provide a parking position for a DUT until the decision to skip or forward it is made.

4.1.4.3 Parallel measurements on multiple DUTs

The support for parallel measurements on multiple DUTs depends on the analyzer type:

- For a 2-port R&S ZNB or a 4-port R&S ZNB without internal second source (see Section 4.7.15, "Second internal generator", on page 258) this feature is **not** available.

- For a 4-port R&S ZNB with Second internal generator, VNA ports 1 and 2 can measure one DUT, while port 3 and port 4 can measure another one (using the same stimulus settings).
- For analyzers with Second internal generator, to reduce "crosstalk" between the DUTs a frequency offset can be applied between the corresponding port groups (see "Parallel measurements with frequency offset" on page 86).



- Parallel Frequency Conversion Measurements require identical conversion settings for all port groups.
- Parallel Intermodulation Measurements are **not** supported.
- Parallel measurements are **not** possible if External switch matrices are used.

Parallel measurements with frequency offset

When performing parallel measurements on a R&S ZNB with Second internal generator, it is possible to specify a minimum frequency offset between the port groups. This is particularly useful for situations where "Crosstalk" between different DUTs would otherwise make the simultaneous measurement impossible (e.g. in wafer prober applications).

Example:

Consider a linear frequency sweep with 101 points from 100 MHz to 200 MHz resulting in a frequency step size of 1 MHz. When two port groups are defined and a minimum frequency offset of 1 MHz is specified, a parallel measurement with frequency offset is performed internally as follows:

- In the first step, port group 1 is not measured; port group 2 is measured at 100 MHz.
- In the second measurement step, port group 1 is measured at 100 MHz; port group 2 is measured at 101 MHz.
- In the third measurement step, port group 1 is measured at 101 MHz; port group 2 is measured at 102 MHz.
- …
- In the 101st measurement step, port group 1 is measured at 199 MHz; port group 2 is measured at 200 MHz.
- Finally, in the 102nd measurement step, port group 1 is measured at 200 MHz; port group 2 is not measured.

Parallel measurement with frequency offset is transparent to the user: All port groups are measured in the requested frequency range. The results are available in the same form as if they were obtained in separate measurements without frequency offset.

During parallel measurements with frequency offset, the firmware uses a modified IF as compared to measurements without frequency offset. Because this modified IF requires a special calibration, it is essential to perform the Calibration with the same Frequency Offset settings as for the actual measurement; otherwise the calibration is turned off (see "Cal Off label" on page 155).

4.1.4.4 Stimulus and sweep types

The function of the Stimulus hardkeys [Start], [Stop], [Center] and [Span] depends on the sweep type.

Table 4-1: Function of Stimulus keys

Sweep type	[Start](unit)	[Stop] (unit)	[Center] (unit)	[Span] (unit)
"Lin Freq"	"Start Frequency" (Hz)	"Stop Frequency" (Hz)	"Center Frequency" (Hz)	"Span Frequency" (Hz)
"Log Freq"	"Start Frequency" (Hz)	"Stop Frequency" (Hz)	–	–
"Segmented"	–	–	–	–
"Power"	"Start Power" (dBm)	"Stop Power" (dBm)	"CW Frequency" (Hz)	"CW Frequency" (Hz)
"CW Mode"	"CW Frequency" (Hz)	"Number of Points" (-)	"CW Frequency" (Hz)	"CW Frequency" (Hz)
"Time"	"CW Frequency" (Hz)	"Stop Time" (s)	"CW Frequency" (Hz)	"CW Frequency" (Hz)

The ranges of numerical values must be compatible with the instrument model. The conditions for the stimulus range depend on the sweep type:

- **"Lin Freq" / "Log Freq" / "Segmented"**
  The supported frequency range depends on the instrument type; for an overview refer to Table 7-18.
  - For "Lin Freq" and "Log Freq" sweeps, if the number of sweep points is greater than 1, the stop frequency must be greater than the start frequency and the span must be ≥ 1 Hz. If a stop frequency smaller or equal than the current start frequency is set, then the start frequency is adjusted and vice versa.
  - For "Segmented" sweeps, the start and stop frequency in a sweep segment must not be different. So with a segmented sweep you can measure n_1 points at frequency f_1 (in segment 1), n_2 points at frequency f_2 (in segment 2) etc.
- **"Power"**
  Start and stop power are both entered in absolute units (dBm). Start and stop power must be different; the stop power must be larger than the start power. If a stop power smaller than the start power is set, then the start power is adjusted automatically and vice versa.
  The power corresponds to the actual source power at the test ports (channel base power P_b). After a port power calibration, this source power is available at the calibrated reference plane.
- **"CW Mode"**
  The stimulus hardkeys define the fixed stimulus frequency ("CW Frequency") and the "Number of Points" of the measurement. The other sweep parameters (e.g. the "Sweep Time") are set via Channel – [Sweep] > "Sweep Params".

- **"Time"**
  The stimulus hardkeys define the fixed stimulus frequency ("CW Frequency") and the total sweep time ("Stop Time") of the measurement. The other sweep parameters (e.g. the "Number of Points") are set via Channel – [Sweep] > "Sweep Params".
  The sweep time is entered in seconds and must be positive.

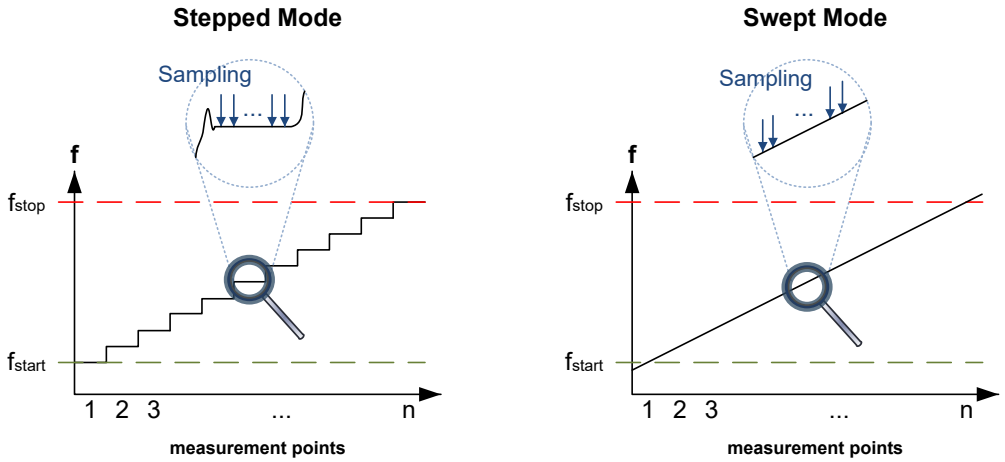
The selected sweep range applies to all source and receive ports of the analyzer.

In arbitrary mode (with option R&S ZNB-K4), you can define port-specific frequencies and powers; see Section 5.12.2.2, "Port Settings dialog", on page 686.

4.1.4.5 Stepped vs. swept mode

The R&S ZNB provides two sweep modes: stepped mode and swept mode.

In stepped mode (default), the frequency is changed stepwise and sampling is performed at a tuned frequency for each measurement point. In swept mode, on the other hand, sampling is performed with the frequency always swept for each measurement point. Since in stepped mode at each measurement point a certain time is required until the frequency becomes stable, sweeps in swept mode are typically much faster than in stepped mode - at the cost of a lower measurement precision.



Apart from the loss in precision, there can be other technical reasons that prevent the swept mode from being advantageous. For instance if the sweep is triggered by events caused by external devices such as part handlers, generators or power meters, the sweep (implemented in hardware) has to be interrupted by the software regularly and cannot proceed autonomously after the initial phase lock.

Swept mode is **not** supported on the following instruments:

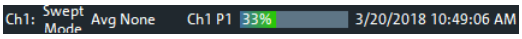
- R&S ZNB43
- R&S ZNBT equipped with synth boards SYNTH13DDS_DS or SYNTH13DDS_L, either ex factory or via upgrade.

  These synth boards are shipped with:

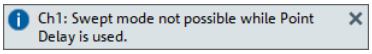
  – R&S ZNBT8|26|40 with serial number ≥ 200000
  – R&S ZNBT20 variant 64 (order no. 1332.9002K64)

For information on the synth boards of your instrument, open the hardware tab of the system info dialog and watch out for the SYNTH components in the "Assemblies" table. See "Hardware tab" on page 829.

If the swept mode is selected and the R&S ZNB actually uses it (for at least one segment), this is indicated in the status bar:



If on the other hand the swept mode is not used although it was selected, the underlying reason is displayed in an information popup:



- The data in the specifications document are stated for **stepped mode**; in swept mode they are not guaranteed.
- Compared to stepped mode the trace noise in swept mode is generally higher.
- DUTs with long group delay cannot be measured correctly in swept mode.
- It is essential to perform the Calibration with the same stepped/swept mode settings as for the actual measurement; otherwise the calibration is turned off (see "Cal Off label" on page 155)

4.1.5 Data flow

The analyzer processes the raw measurement data in a sequence of stages to obtain the displayed trace. The following subsections give an overview.

4.1.5.1 Channel data flow

The diagram below illustrates the data processing stages for the entire channel. All stages are configurable.

Note that the channel data flow for S-parameters (and quantities derived from S-parameters such as impedances, admittances, stability factors) differs from the channel data flow for wave quantities (and derived quantities such as ratios).

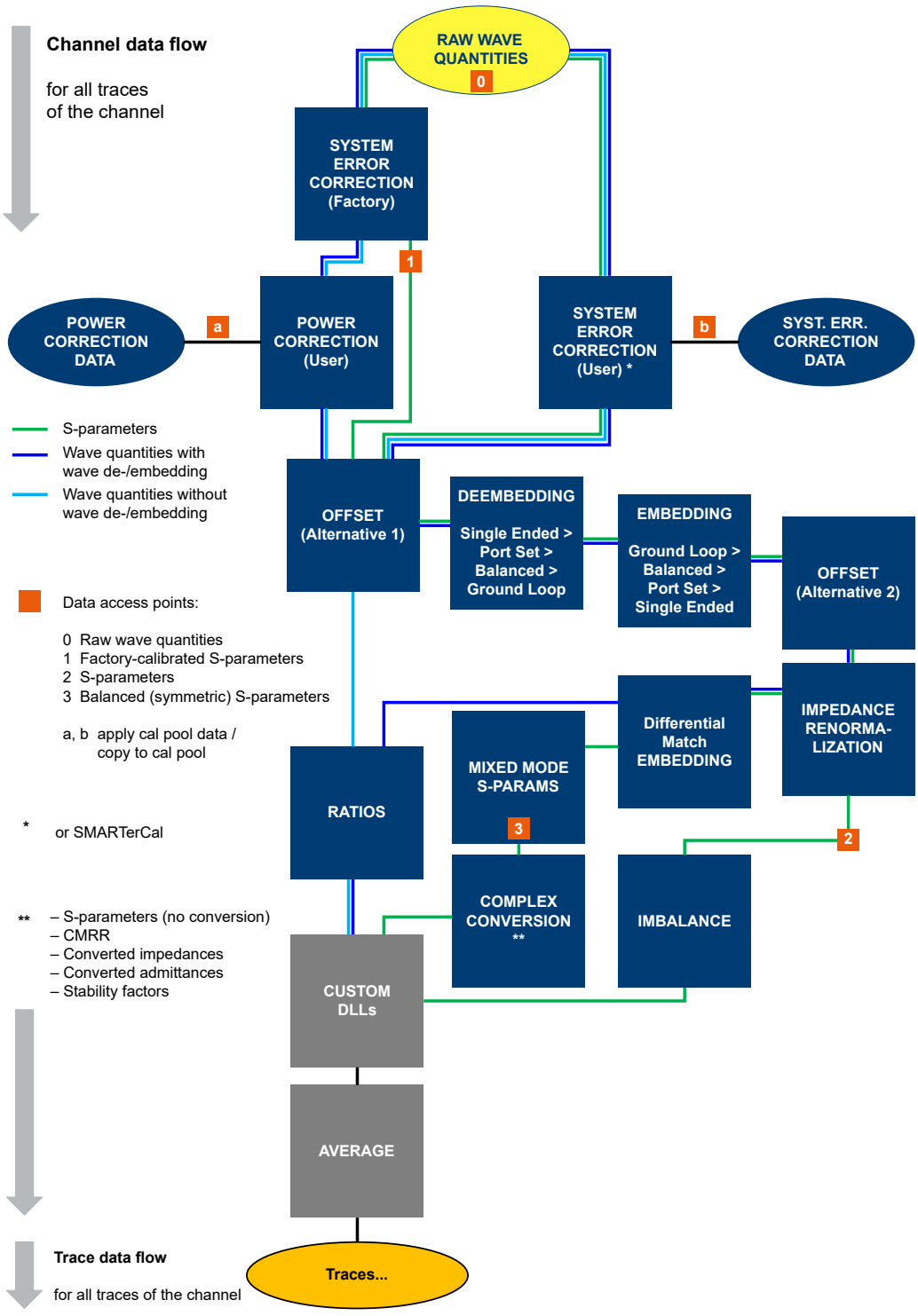


Figure 4-3: Channel data flow

Wave de-/embedding

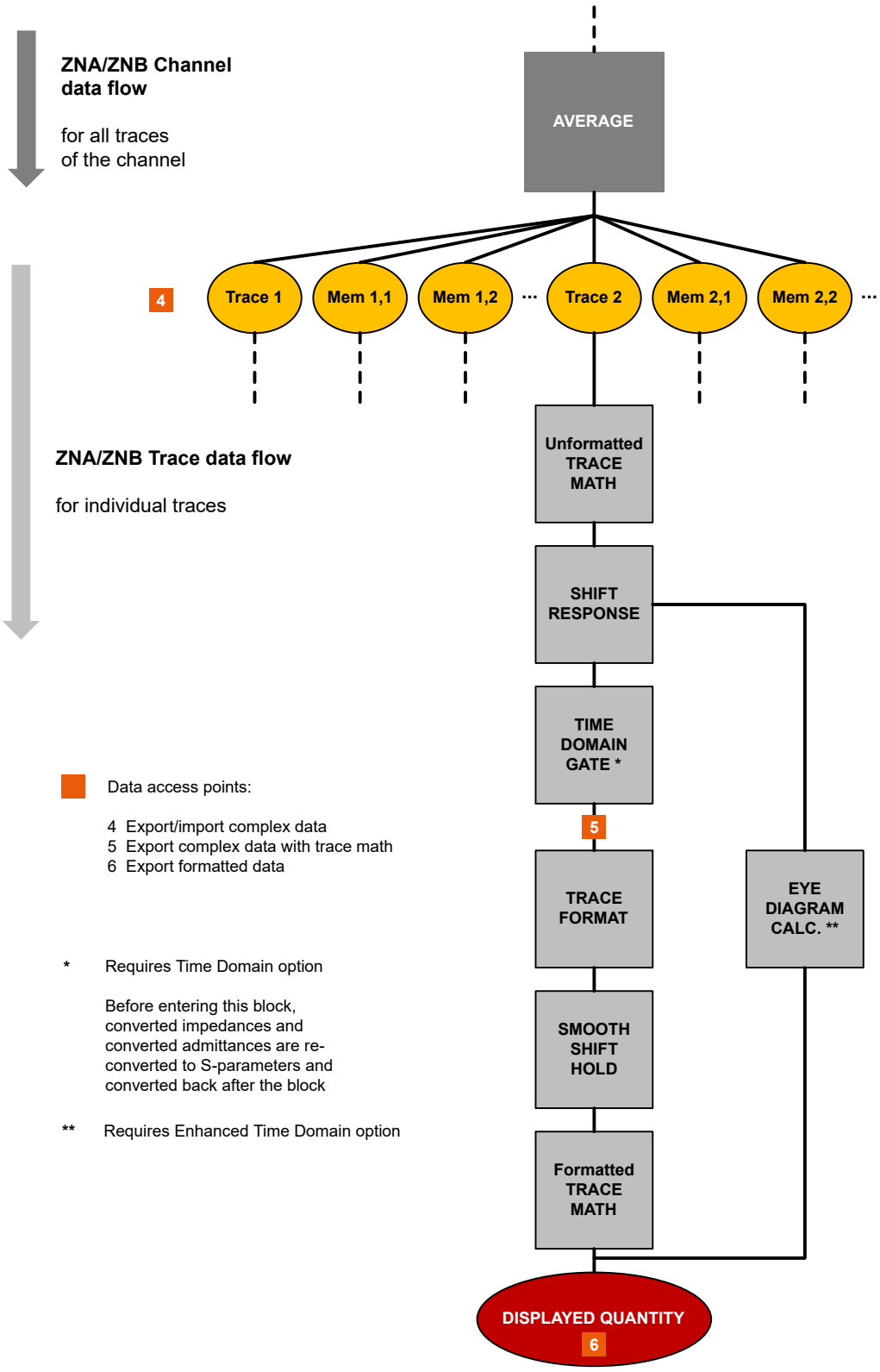
In its original implementation, de-/embedding was calculated on the S-parameter branch of the channel calculation chain. After applying factory calibration and/or user

calibration on the measured wave quantities, intermediate results were transformed to the S-parameter domain. Before each de-/embedding step, the S-parameters were transformed to conductance (~ wave quantities), and vice versa afterwards.

The new "wave de-/embedding" remains in the wave quantity (~conductance) domain and calculates all de-/embedding steps in one multiplication step, without unnecessary matrix inversions. With "wave de-/embedding", the computation time does not depend on the number of embedding and/or de-embedding steps but is constant.

4.1.5.2 Trace data flow

The diagram below illustrates the processing stages the channel measurement data run through for each individual trace. Again, all stages are configurable.


ZNA/ZNB Channel data flow
for all traces of the channel
AVERAGE
4
Trace 1
Mem 1,1
Mem 1,2
Trace 2
Mem 2,1
Mem 2,2
ZNA/ZNB Trace data flow
for individual traces
Unformatted TRACE MATH
SHIFT RESPONSE
TIME DOMAIN GATE *
5
TRACE FORMAT
EYE DIAGRAM CALC. **
SMOOTH SHIFT HOLD
Formatted TRACE MATH
DISPLAYED QUANTITY
6
Data access points:
4 Export/import complex data
5 Export complex data with trace math
6 Export formatted data
* Requires Time Domain option
Before entering this block, converted impedances and converted admittances are re-converted to S-parameters and converted back after the block
** Requires Enhanced Time Domain option

4.2 Screen elements

This section describes manual operation of the analyzer, including trace settings, markers and diagrams. For a description of the different quantities measured by the instrument, refer to Section 4.3, "Measurement results", on page 116.

4.2.1 Display elements of a diagram

The central part of the screen is occupied by one or more diagrams.

A diagram is simply a rectangular portion of the screen used to display traces. Diagrams are independent of trace and channel settings. A diagram can contain a practically unlimited number of traces which can be assigned to different channels.

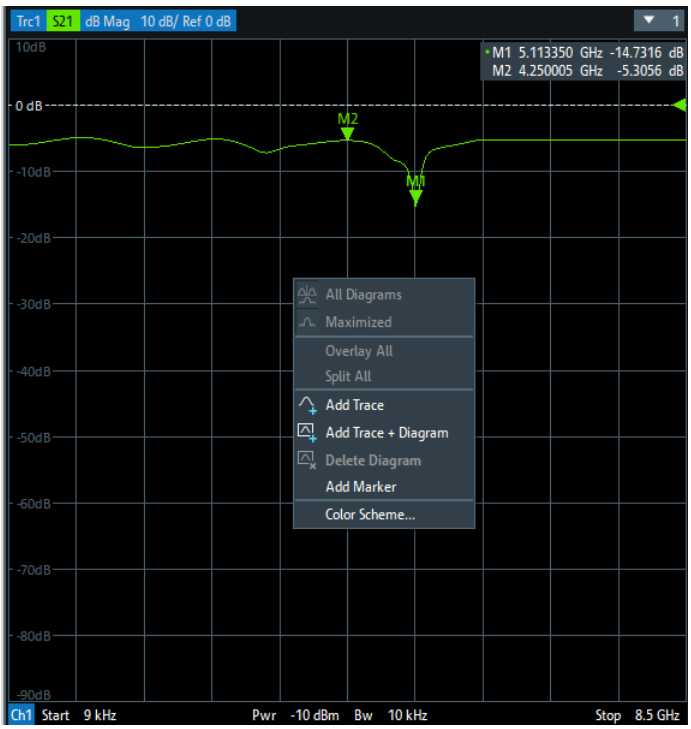
Most diagram settings are arranged in the "Display" softtool (hardkey System – [Display]). To assign traces and channels to diagrams, use the control elements on the "Trace Config" > "Traces" and "Channel Config" > "Channels" softtool tabs (hardkeys Trace – [Trace Config] and Channel – [Channel Config]).

Diagrams can contain:

- A title (optional)
- The diagram number (or label)
- Measurement results, in particular traces and marker values (optional)
- An indication of the basic channel and trace settings
- Context menus providing settings which are related to a particular display element
- Error messages



The examples in this section have been taken from Cartesian diagrams. All other diagram types provide the same display elements.



4.2.1.1 Title

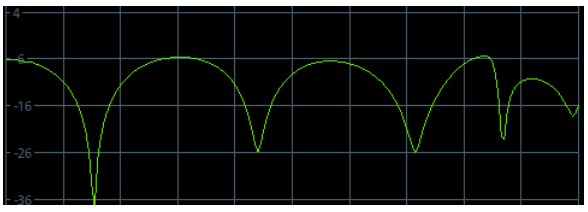
An optional title across the top of the diagram can be used for a brief description of the diagram contents.



Select System – [Display] > "Diagram" > "Title" to enter the diagram title and "Show Title" to display or hide it.

4.2.1.2 Traces

A trace is a set of data points displayed together in the diagram. The individual data points are connected so that each trace forms a continuous line.



The trace can be complemented by the following display elements, plotted with the same color:

- Reference value (for all traces): The reference value is indicated with a triangle at the right edge of the diagram and a dashed, horizontal line. The value and position

of the triangle can be changed to modify the diagram scale and shift the trace vertically.

- Measured quantity (for the active trace): The measured quantity is indicated in the trace list; see "Trace list and trace settings" on page 95.

A trace can be either a data trace, a memory trace, or a mathematical trace; see "Trace types" on page 95.

Trace types

The analyzer uses traces to display the current measurement result in a diagram. It is also capable of storing traces to the memory, recalling stored traces, and defining mathematical relations between different traces. There are three basic trace types:

- Data traces show the current measurement data and are continuously updated as the measurement goes on. Data traces are dynamic traces.
- Memory traces are generated by storing the data trace to the memory. They represent the state of the data trace at the moment when it was stored. Memory traces are static traces which can be stored to a file and recalled.
- Mathematical traces are calculated according to a mathematical relation between constants and the data or memory traces of the active recall set. A mathematical trace that is based on the active data trace is dynamic.

It is possible to generate an unlimited number of memory traces from a data trace and display them together. Markers and marker functions are available for all trace types.

The type of each trace in a diagram is indicated in the trace list: "MEM<no>" at the beginning of the trace name indicates a memory trace (with default naming), Math at the end of the trace label indicates a mathematical trace. You can also hide a trace ("Invisible") without deleting it.



Trace list and trace settings

The main properties of all traces assigned to the diagram are displayed in the trace list in the upper part of the diagram.



Each line in the trace list describes a single trace. The active trace is highlighted ("Trc5" in the example above). The lines are divided into several sections with the following contents (from left to right):

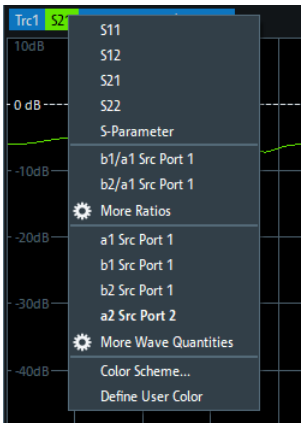
- The **trace name** appears in the first section. The default names for new traces are Trc<n> with n automatically selected. A "Mem" at the beginning of the trace name indicates a memory trace (default naming). To change the trace names, open the "Trace Manager" from any trace name segment's context menu.
- The **measured quantity** (e.g. an S-parameter or a ratio) appears on a colored background. The source port for wave quantities and ratios is indicated in brackets.

- The **format** section shows how the measured data is presented in the graphical display. Use the context menu of the format section to change the format of the related trace.
- The next sections show the value of the vertical or radial diagram divisions ("Scale Div.") and the reference value ("Ref").
- The **channel** section shows the channel that each trace is assigned to. It is omitted if the all traces in the diagram are assigned to the same channel.
- The **type** section indicates "Invisible" if a trace is hidden and Math if the trace is a mathematical trace. "Gat" indicates that a time gate is active for the trace. Use the "Mem Math" and "Traces" tabs of the "Trace Config" softtool to display and hide data and memory traces, and to define mathematical traces.

- The respective section's context menu (except for the type section) provides access to the most common related tasks.
- If the size of the diagram is too small, some of the sections are hidden. Enlarge or maximize the diagram to display all sections.

Example:

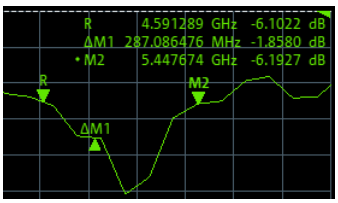
The following context menu is assigned to the measured quantity section in the trace list:



A label "Cal Off" appears at the end of the trace line if the system error correction no longer applies to the trace.

4.2.1.3 Markers

Markers are tools for numerical readout of measured data and for selecting points on the trace, or, in general, in the diagram area. A marker is displayed with a symbol (e.g. a triangle, a crossbar or a line) on the trace, which can be a data trace or a memory trace. At the same time, the coordinates are displayed in a marker info field or in a table. Each marker can be defined as a normal marker (M), reference marker (R), or delta marker (ΔM).



- A normal marker ("M1, M2") determines the coordinates of a measurement point on the trace.
- The reference marker ("R") defines the reference value for all delta markers.
- A delta marker ("DeltaM1, DeltaM2") indicates the coordinates relative to the reference marker.

A special set of markers M1 to M4 is provided for bandfilter search mode.

The most common tasks to be performed with markers can be achieved using the "Marker" menu functions:

- Determine the coordinates of a measurement point on the trace. In polar diagrams where no x-axis is displayed, markers can be used to retrieve the stimulus value of specific points.
- Determine the difference between two trace points or the relative measurement result ("Delta Mode").
- Convert a complex measurement result into other formats.

Markers also play an important role in performing the following advanced tasks:

- Change the sweep range and the diagram scale ("Marker Function").
- Search for specific points on the trace ("Marker Search", "Target Search", "Bandfilter").

Activating and moving markers

To activate a marker, either select the marker symbol itself or the corresponding line in the marker info field.

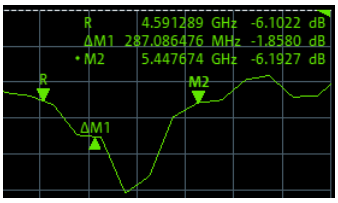
To move the active marker on the trace, use one of the following methods:

- Drag the marker symbol to the desired position (Cartesian diagrams only).
- Activate the "Markers" tab of the "Marker" softtool (Trace – [Marker]) and enter the related stimulus value numerically.
- Use the functions on the "Marker Search" softtool tab to move the marker to a specific position.

If the marker position is adjusted using the roll key, the mouse or the cursor keys, it always remains within the sweep range. If set explicitly by entering a numeric value, the marker position can be outside the sweep range. In this case, the marker symbol is automatically positioned to the start or stop value of the sweep range, whichever is closer.

Marker info field

The coordinates of all markers defined in a diagram can be displayed in the info field, which by default is located in the upper right corner.



The info field contains the following information:

- "M1, M2" denote the marker numbers. Markers are displayed with the same color as the associated trace.
- The marker coordinates are expressed in one of the marker formats selected via Trace – [Marker] > "Marker Props" > "Marker Format". The formats of the markers assigned to a trace are independent of each other and of the trace format settings.
- The active marker has a dot placed in front of the marker line.
- "R" denotes the reference marker. A "Δ" sign placed in front of the marker line indicates that the marker is in delta mode.

Open the context menu of the marker info field to access frequently used marker settings.



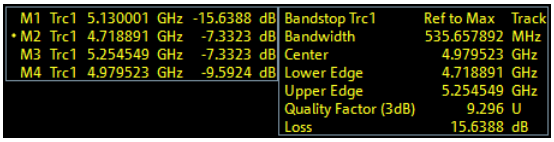
Customizing the marker info field

To change the position, appearance or contents of the marker info field, use one of the following methods:

- The info field can be moved to several positions in the upper and lower part of the active diagram. Drag & drop it to the desired position.
- To change the format of the active marker, select [TRACE] > "Marker" > "Marker Properties" > "Marker Format".
- To express the coordinates of the active marker relative to the reference marker, activate the delta mode [TRACE] > "Marker" > "Marker Properties" > "Delta Mode".

Info Table

If you wish to reserve the full diagram space for traces, you can drag & drop the marker info field to the info table.



The info table is hidden by default. To display it, open the "Display" softtool (System – [Display]), activate its "Config" tab and select "Info Table" – "Show".

Marker format

Marker values can be formatted according to the current trace format, according to the default marker format of the related trace (Trace > [Format] > "Format" > "Dflt Marker Frmt"), or formatted individually (Trace > [Marker] > "Marker Props" > "Marker Format").

The available marker formats are defined for all measured quantities and trace formats (see Section 4.2.3.3, "Measured quantities and trace formats", on page 116). Essentially, a marker format is simply a conversion between points on a complex-valued trace (the raw measurement data) and the respective target format. This must be kept in mind when interpreting the results and physical units displayed.

The following table describes how a complex marker value z = x + jy is converted. It makes use of the polar representation $z = x + jy = |z|\ e^{j\varphi(z)}$, where

$|z| = (x^2 + y^2)^{1/2}$ and $\varphi(z) = \arctan(y / x)$

Table 4-2: Marker formats

Marker Format	Description	Formula
Default	• For an **individual marker**, this means that the marker is formatted according to the default marker format of the related trace. • For a **trace's default marker format**, this means that the default format is (dynamically) adjusted according to the selected trace format.	–
Lin Mag	Magnitude of z, unconverted	$\|z\| = \text{sqrt}(x^2 + y^2)$
dB Mag	Magnitude of z in dB	dB Mag(z) = 20 * log|z| dB
Phase	Phase of z	$\varphi(z) = \arctan(y/x)$
Delay	Group delay, neg. derivative of the phase response*)	$-d\varphi(z) / d\omega$, where $\omega = 2\pi * f$ with stimulus frequency f
Delay Derivation	Derivative of the delay (= second derivative of the phase response)	$-d^2\varphi(z) / d\omega^2$
Real	Real part of z	Re(z) = x
Imag	Imaginary part of z	Im(z) = y
SWR	(Voltage) Standing Wave Ratio	SWR = (1 + |z|) / (1 – |z|)

Marker Format	Description	Formula
dB Mag Phase	Magnitude of z in dB and phase in two lines	20 * log|z| dB arctan (Im(z) / Re(z))
Lin Mag Phase	Magnitude of z (unconverted) and phase in two lines	|z| arctan (Im(z) / Re(z))
Real Imag	Real and imaginary part of z in two lines	x y
R + j X or R + j X series***)	(Series) impedance: Unnormalized (series) resistance, reactance, and either inductance or capacitance, in three lines (Smith diagram)	R_s X_s L_s or C_s**)
R + j X parallel***)	Parallel impedance: Unnormalized parallel resistance, reactance, and either inductance or capacitance, in three lines (Smith diagram)	R_p X_p L_p or C_p**)
G + j B or G + j B series***)	(Series) admittance: Unnormalized (series) conductance, susceptance, and either inductance or capacitance, in three lines (Inverted Smith diagram)	G_s B_s L_s or C_s**)
G + j B parallel***)	Parallel admittance: Unnormalized parallel conductance, susceptance, and either inductance or capacitance in three lines (Inverted Smith diagram)	G_p B_p L_p or C_p**)
Imp Mag or IMP Mag series***)	Magnitude of (series) impedance**)	$\|Z_s\| = (R_s^2 + X_s^2)^{1/2}$
Imp Mag parallel***)	Magnitude of parallel impedance**)	$\|Z_p\| = (R_p^2 + X_p^2)^{1/2}$
Adm Mag or Adm Mag series***)	Magnitude of (series) admittance**)	$\|Y_s\| = (G_s^2 + B_s^2)^{1/2}$
Adm Mag parallel***)	Magnitude of parallel admittance**)	$\|Y_p\| = (G_p^2 + B_p^2)^{1/2}$
Index	Index of the current sweep point	–
Noise	Noise power density measured at the marker position (1 Hz bandwidth).	Cum On, Feel the Noize ...

*) The delay aperture is defined in the [Trace Config] > "Format" softtool.

**) An impedance Z is represented as Z = R + jX, the corresponding admittance as Y = 1/Z = G + jB. For X ≥ 0, we have an inductance L = X/ω, for X < 0 we have a capacitance C = 1/(ωX), where ω denotes the stimulus frequency.

***)Only available for transmission measurements (see Section 4.3.3.1, "Converted impedances", on page 121)

Marker coupling

It connects the markers of a set of traces.

Marker coupling allows you to compare different measurement results (assigned to different traces) at the same stimulus value. It connects the markers of a set of traces.

Marker coupling can be enabled:

- Either for all traces in the active recall set that have the same stimulus variable as the active trace
- or for all traces in a channel
- or for all traces in a diagram that have the same stimulus variable as the active trace

When marker coupling is enabled, the same markers are activated for all related traces: if a marker was active for some related trace, then it is activated for all related traces.

While marker coupling is active, the marker sets of the related traces are always kept in sync, i.e.:

- If a marker is added to (removed from) one of the related traces, it is also added to (removed from) the other related traces.
- If a marker is moved to a particular stimulus value for one of the related traces, then it is moved to this stimulus value for all related traces.
  If the new stimulus value is outside a trace's sweep range, the marker value is invalid for this trace. The corresponding info field only displays the stimulus value.



Only one kind of marker coupling can be enabled. For instance, it is not possible to couple markers per channel and per diagram at the same time.

Basic marker search functions

The search functions are tools for searching measurement data according to specific criteria. A search consists of analyzing the measurement points of the current trace (or of a user-defined subrange termed the "Search Range") to find one of the following:

- Absolute or relative (local) maxima and minima (peak search).
- Trace points with a specific response value (target search).
- Trace segments with a shape that is characteristic for bandpass or bandstop filters (bandfilter search); see "Bandfilter search" on page 102.

When the search is activated, the active marker is moved to the (next) point that meets the search criteria. If the trace contains no markers, a marker M1 is created and used for the search. The search result is displayed in the marker info field. If no search result can be found, the marker remains at its original position.

Some search functions can be activated repeatedly to find all possible search results. Moreover the analyzer provides a "Tracking" mode where the search is repeated after each sweep.

Multiple peak search

Multiple peak search allows you to find multiple local minima/maxima at once. Markers 1 to 10 are assigned to the peaks detected from the start frequency towards the stop frequency. Multiple peak search uses its own search and tracking settings; search and tracking settings for standard marker search are ignored.

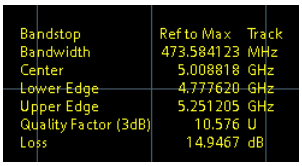
Bandfilter search

In a bandfilter search, the R&S ZNB locates trace segments with a bandpass or bandstop shape and determines characteristic filter parameters.

Bandpass and bandstop regions can be described with the same parameter set:

- A bandpass region contains a local maximum around which the magnitude of the trace falls off by more than a specified value.
- A bandstop region contains a local minimum around which the magnitude of the trace increases by more than a specified value.

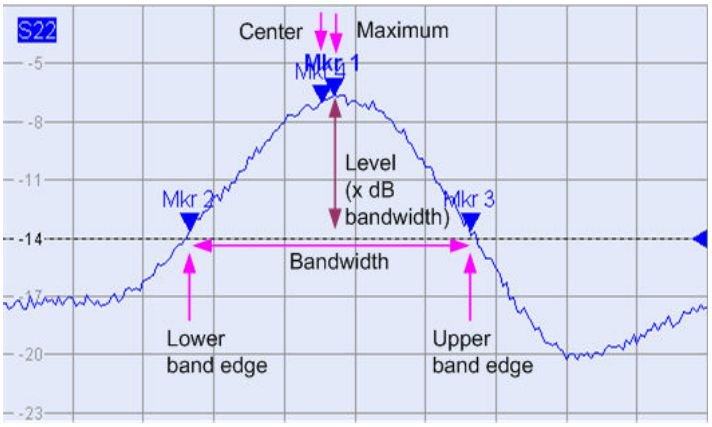
The analyzer locates bandpass and bandstop regions and determines their position ("Center" frequency) and shape ("Bandwidth", "Lower Edge" and "Upper Edge", quality factor. For a meaningful definition of the bandwidth factor, the trace format must be "dB Mag".



The info field contains the following search results:

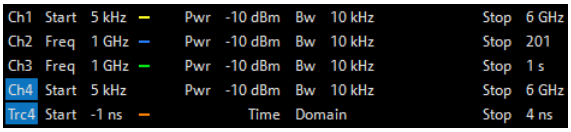
- "Bandwidth" is the n-dB bandwidth of the bandpass/bandstop region, where n is a selectable bandwidth factor. The bandwidth is equal to the difference between the lower and the upper band edge frequency.
- "Center" is calculated as the geometric or arithmetic mean of the lower band edge frequency f_{LBE} and the upper band edge frequency f_{UBE}:
  $f_{Center} = sqrt (f_{LBE} * f_{UBE})$ (geometric mean) or
  $f_{Center} = 1/2 (f_{LBE} + f_{UBE})$ (arithmetic mean)
  The arithmetic mean is always higher than the geometric mean. The values are close if the bandwidth is small compared to the geometric mean of the band edges.
- "Lower Edge" is the closest frequency below the maximum (or minimum), where the trace value is equal to the maximum (minimum) value minus (plus) n dB.
- "Upper Edge" is the closest frequency above the maximum (or minimum), where the trace value is equal to the maximum (minimum) value minus (plus) n dB.
- The "Quality Factor (3 dB)" is the ratio between the "Center" frequency and the 3-dB "Bandwidth"; it does not depend on the selected bandwidth factor.
- The "Quality Factor (BW)" is the ratio between the "Center" frequency and the "Bandwidth" displayed above. This result is available only if the selected bandwidth factor is different from 3 dB.

- "Loss" is the loss of the filter at its maximum (or minimum) and is equal to the response value of marker no. 1. For an ideal bandpass filter, the loss is zero (0 dB), for an ideal bandstop filter it is –∞ dB.



4.2.1.4 Channel list and channel settings

The main properties of all channels assigned to the traces in the diagram are displayed in the channel list below the diagram.



Each line in the channel list describes a single channel. The channel of the active trace is highlighted. The lines are divided into several sections with the following contents (from left to right):

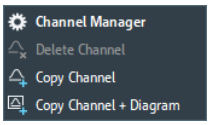
- The "Channel Name" appears in the first section. The default names for new channels are Ch<n> with an automatically assigned number <n>. If a time domain transform is active, the R&S ZNB displays an additional line to indicate the stimulus range of the displayed time-domain trace.
  Open the "Channel Manager" from the name segment's context menu to change the channel name.
- The **measurement mode identifier** section (optional) indicates a special test mode of the channel, e.g. the measurement at arbitrary port frequencies ("Arb Port n").
- **Start** indicates the lowest value of the sweep variable (e.g. the lowest frequency measured), corresponding to the left edge of a Cartesian diagram.
- The **color legend** shows the display color of all traces assigned to the channel. The colors are different, so the number of colors is equal to the numbers of traces assigned to the channel.
- The values behind the color legend show the **constant stimulus value**, which is either the power of the internal signal source (for frequency sweeps and time sweeps) or the CW frequency (for power sweeps), and the measurement bandwidth ("BW").

- **Stop** indicates the highest value of the sweep variable (e.g. the highest frequency measured), corresponding to the right edge of a Cartesian diagram.

Open a segment's context menu to access common related tasks.

Example:

The following context menu is assigned to the channel name section:



The settings in the context menus correspond to the most common functions in the Channel – [Channel Config] > "Channels" softtool tab, the "Stimulus" softtool (opened via Stimulus hardkeys), the Channel – [Sweep] > "Sweep Params" softtool tab, and the Channel – [Pwr Bw Avg] softtool.

4.2.1.5 Context menus

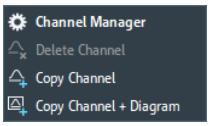
To provide access to the most common tasks and speed up the operation, the analyzer offers context menus (right-click menus) for the following display elements:

- Diagram
- Marker info field
- Trace list (separate context menus for trace name section, measured quantity section, format section, scale section, and channel section)
- Channel list (separate context menus for channel name section, sweep range section, additional parameter section)

To open a context menu associated with a display element, tap and hold or right-click the element for some seconds.

Example:

The following context menu is assigned to the channel name section in the channel list:

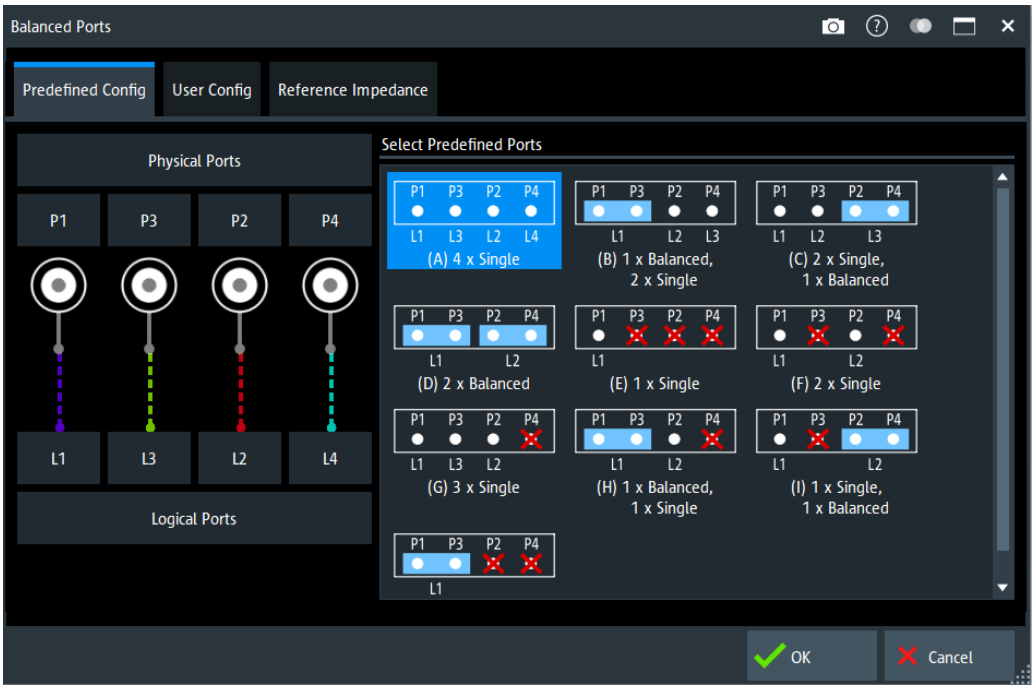


The functions of the context menu can also be called using the menu bar or the related softtool panels. Use whatever method is most convenient.

4.2.2 Dialogs

Dialogs provide groups of related settings and allow to make selections and enter data in an organized way. All softkeys with three dots behind their labeling (as in "Balanced

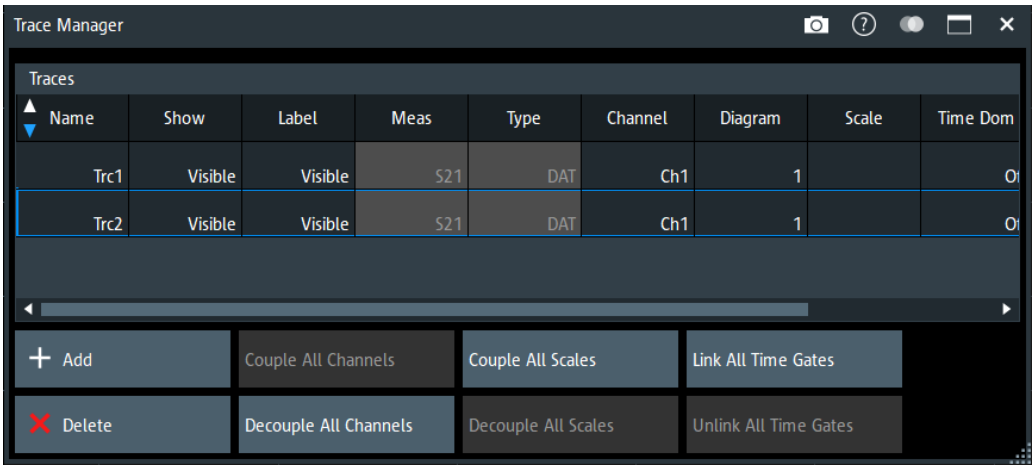
Ports") call up a dialog. The dialogs of the analyzer have an analogous structure and several common control elements.



Dialogs are controlled in the usual way. For an introduction, refer to Section 3.3.4, "Working with dialogs", on page 56.

4.2.2.1 Immediate vs. confirmed settings

In some dialogs settings take effect immediately, so that the effect on the measurement is observable while the dialog is still open. This behavior is particularly useful when a numeric value is incremented or decremented, or when display elements are added or removed.



In most dialogs, however, it is possible to cancel an erroneous input before it takes effect. The settings in such dialogs must be confirmed explicitly.

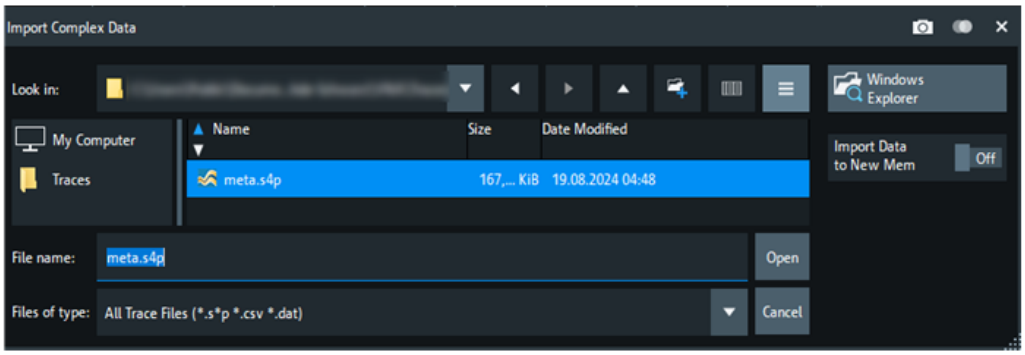
The two types of dialogs are easy to distinguish:

- Dialogs with immediate settings provide a "Close" button but no "OK" button.
  Example: "Trace Manager" dialog
- Dialogs with confirmed settings provide both an "OK" button and a "Cancel" button.
  Example: "Balanced Ports" dialog

Immediate settings can be undone using [Undo].

4.2.2.2 Common dialogs

Open Dialog
The "Open File" dialog is used to open various file types (cal kit data, limit lines, sweep segment lists, ...).

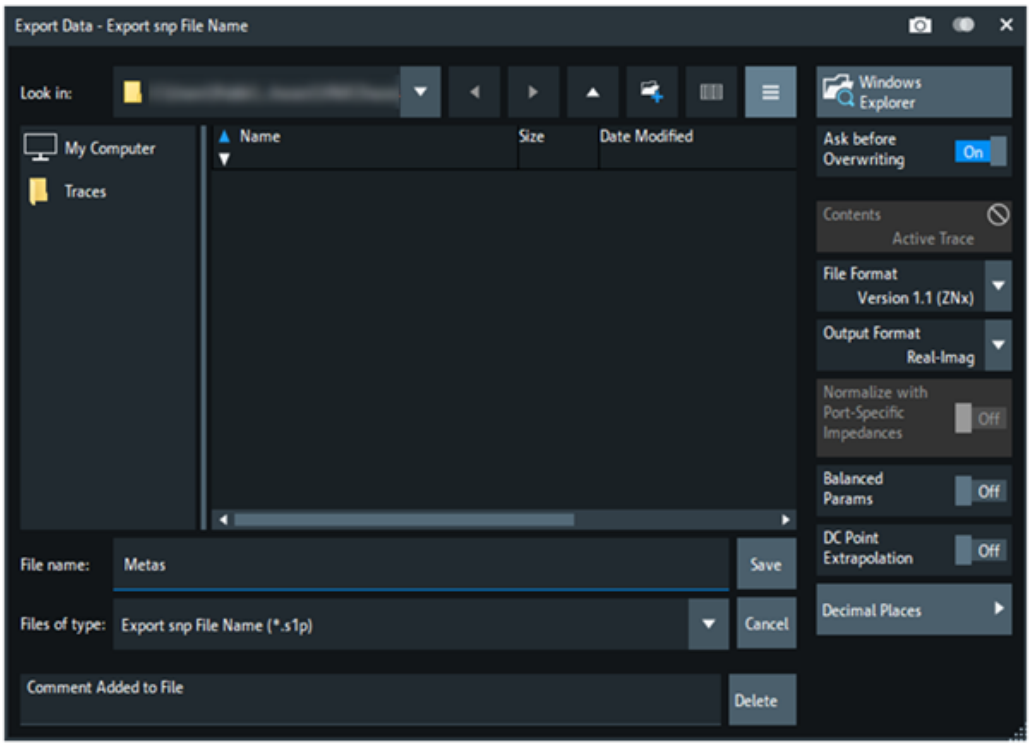


Depending on the context, the dialog is displayed with different caption, default directory ("Traces" in the above screenshot), and file type filters. Context-specific options ("Import Data to New Mem" in the above screenshot) are accessible via controls in the section below the "Windows Explorer" button.

- "Look in:" specifies the directory to be listed. The icons to the right of the pull-down list are provided for easy navigation in the file system (place the cursor on the icons to obtain "Whats this" help).
- "Windows Explorer" opens the selected directory in the Windows Explorer.
- "File name" specifies a filename to save the current data. The analyzer adds the extension in the "Files of type" field.
- "Files of type" filters the displayed files by type.
- "Open" opens selected file and closes the dialog.
- "Cancel" closes the dialog without opening a file.

Tip: Dialog properties (e.g. the current directory) are remembered when the dialog is closed. To restore default directories, select "Use Default Directories" in the Presets tab of the "System Config" dialog.

Save Dialog
The "Save" dialog is used to store various data types (e.g. cal kit data, limit lines, sweep segment lists, ...).



Depending on the context, the dialog is displayed with different caption, default directory ("Traces" in the above screenshot), and file types. Context-specific options (e.g. "Output Format" in the dialog above) are accessible via controls in the section below the "Ask Before Overwriting" toggle button.

- "Look in" specifies the drive and directory in which the data is stored. The icons to the right of the pull-down list are provided for easy navigation in the file system (place the cursor on the icons to obtain "Whats this" help).
- "File name" specifies a filename to save the current data. The analyzer adds the extension in the "Files of type" field.
- "Files of type" selects a particular file type for the created file.
- "Save" saves the data in the selected file and directory and closes the dialog.
- "Cancel" closes the dialog without saving the data.
- "Windows Explorer" opens the selected directory in Windows Explorer.
- If "Ask Before Overwriting" is enabled, overwriting an existing file has to be confirmed.

Tip: Dialog properties (e.g. the current directory) are remembered when the dialog is closed. To restore default directories, select "Use Default Directories" in the Presets tab of the "System Config" dialog.

4.2.2.3 Multi-channel setup dialog

Depending on the desired DUT and measurement type, the measurement channel has to be prepared in a particular way. The multi-channel setup dialog was invented to make this task as easy as possible.

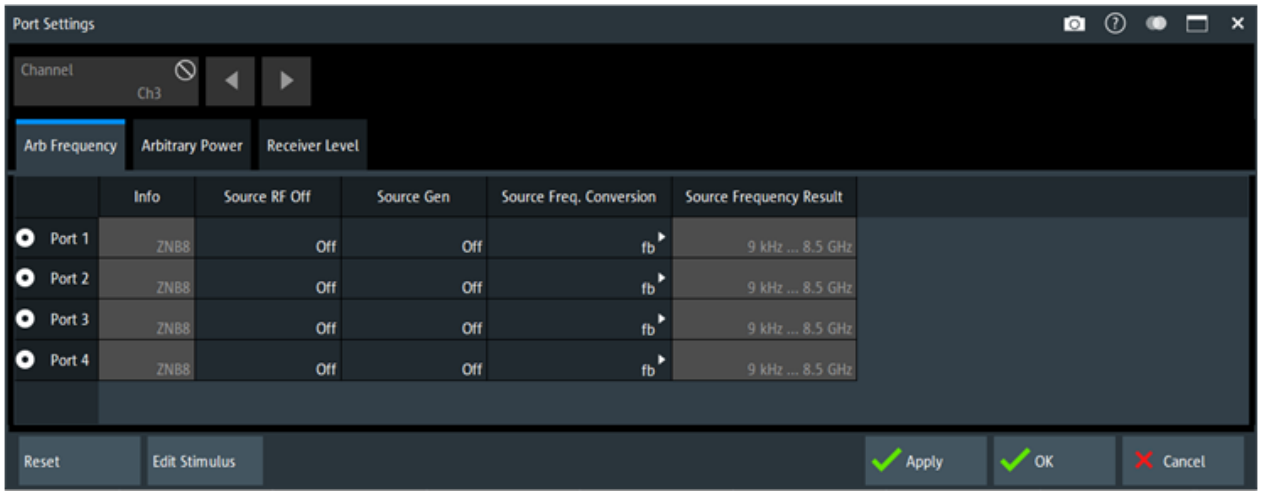


Figure 4-4: Multi-channel setup dialog

The channel at the bottom of the dialog allows you to select the target channel.

"OK" is only active, if the configured measurement setup is valid. Error and warning messages are displayed to point out possible configuration problems.

4.2.3 Trace formats

A trace format defines how a trace is represented in a diagram.

The R&S ZNB supports the following trace formats:

- Cartesian trace formats "dB Mag", "Phase", "SWR", "Unwr Phase", "Lin Mag", "Log Mag", "Real", "Imag", "Delay", and "Delay Derivation".
- Complex trace formats "Polar", "Smith" and "Inv Smith"

The VNA firmware allows arbitrary combinations of trace formats and measured quantities. However, to extract useful information from the measured data, it is important to select a trace format which is appropriate for the analysis of a particular measured quantity; see Section 4.2.3.3, "Measured quantities and trace formats", on page 116.

4.2.3.1 Cartesian trace formats

Cartesian trace formats assign a scalar response to the stimulus value (frequency, power, or time). The response can be calculated from the measured quantity at the related stimulus value, but it can also be the result of some mathematical transformation of the original (unformatted) trace.

Diagram representation

When a Cartesian trace is assigned to a diagram, the stimulus variable appears on the horizontal axis (x-axis), the response values appear on the vertical axis (y-axis).

Graph Scaling

- Except for the "Log Mag" format, the y-axis scale is always linear.
- The x-axis scaling depends on the sweep type of the channel to which the trace is assigned:
  - For sweep types "Lin Freq", "Power", "CW Mode" and "Time" it is scaled linearly.
  - For sweep type "Log Freq", it is scaled logarithmically.

The resulting linear or lin-log grid is plotted with the formatted trace.

The following examples show "dB Mag" Cartesian traces for the same measured quantity and sweep range, but with "Lin Freq" and "Log Freq" sweep types.

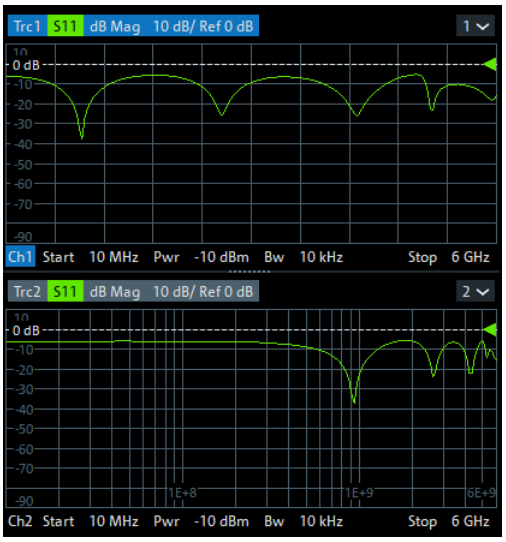


Figure 4-5: S11 trace in dB Mag format: sweep type Lin Freq (top) and Log Freq (bottom)

Conversion of complex to real quantities

Among the measured quantities the R&S ZNB supports, only "Stability" factors, "Power Sensor" results, and "DC" values are real. All other results measured quantities complex.

The following table shows how "real" response values are calculated from complex measurement values z = x + j y (where x, y, z are functions of the sweep variable). The formulas also hold for real measurement values (y = 0).

Trace Format	Description	Formula
"dB Mag"	Magnitude of z in dB	dB Mag(z) = 20 * log|z| dB
"Phase"	Phase of z	φ (z) = arctan (y/x)
"SWR"	(Voltage) Standing Wave Ratio	SWR = (1 + |z|) / (1 – |z|)
"Lin Mag"	Magnitude of z, unconverted	|z| = sqrt (x^2 + y^2)
"Real"	Real part of z	Re(z) = x
"Imag"	Imaginary part of z	Im(z) = y

Trace Format	Description	Formula
"Delay"	Group delay, neg. derivative of the phase response	$\tau_g(z) = -d\varphi(z) / d\omega$ ($\omega = 2\pi * f$ with stimulus frequency f)
"Delay Derivation"	Derivative of the "Delay", i.e. the (negative) second derivative of the "Phase").	$d\tau_g(z)/d\omega = -d^2\varphi(\omega) / d\omega^2$

An extended range of formats and conversion formulas is available for markers. To convert any point on a trace, create a marker and select the appropriate marker format. Marker and trace formats can be selected independently.

4.2.3.2 Complex trace formats

Complex trace formats assign a complex response to the stimulus value (frequency, power, or time).

In diagrams, the response values are always represented as points in the two-dimensional complex plane:

- The complex 0 is located at the center of the diagram.
- The real part is drawn in horizontal direction, the imaginary part in vertical direction.

Result values for consecutive stimulus values are interconnected by straight lines, so the trace is represented as a polygonal chain in the complex plane.

The stimulus axis is not visible. However, the stimulus value for a given trace point can be displayed using a marker.

The difference between the different complex trace formats (Polar,Smith and Inv Smith) is the coordinate system that is used for the representation of the response values and that is graphically overlaid to the formatted trace.

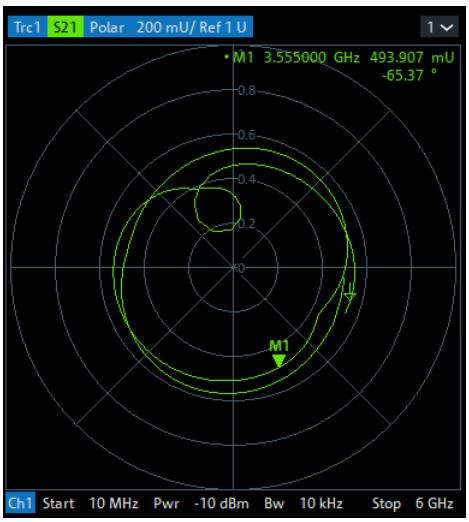
Polar

For "Polar" traces the complex response values are represented in polar coordinates: magnitude and phase.

In a diagram the grid lines overlaid to the trace correspond to points of equal magnitude and phase:

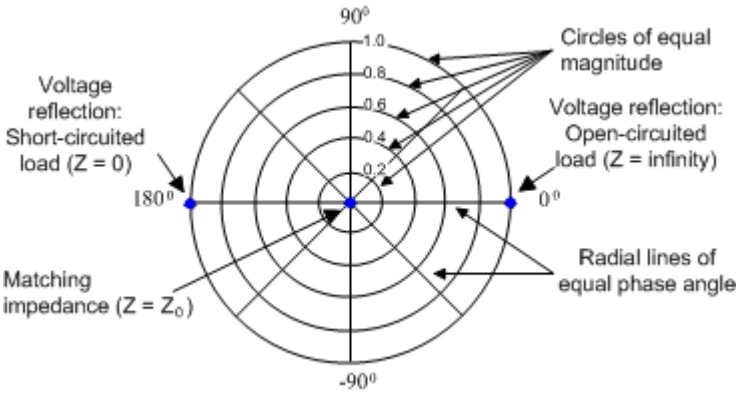
- Points with equal magnitude are located on circles around the complex 0 that is located at the center of the diagram.
- Points with the equal phase are located on straight lines originating at the center.

The following example shows a polar diagram with a marker used to display a pair of stimulus and response values.



Example: Reflection coefficients in polar diagrams

If the measured quantity is a complex reflection coefficient (S_{11}, S_{22} etc.), then the center of the polar diagram corresponds to a perfect load Z_0 at the input test port of the DUT (no reflection, matched input). The outer circumference ($|S_{ii}| = 1$) represents a totally reflected signal.



Examples for definite magnitudes and phase angles:

- The magnitude of the reflection coefficient of an open circuit (Z = infinity, I = 0) is one, its phase is zero.
- The magnitude of the reflection coefficient of a short circuit (Z = 0, U = 0) is one, its phase is –180 deg.

Smith

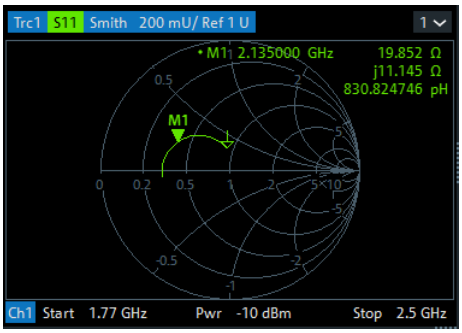
For "Smith" traces the response values are interpreted as reflection coefficients S_{ii} and represented in terms of their corresponding complex impedance $Z(S_{ii}) = R(S_{ii}) + j\,X(S_{ii})$.

In a diagram, the grid lines overlaid to a "Smith" trace correspond to points of equal resistance R and reactance X:

- Points with the same resistance are located on circles.

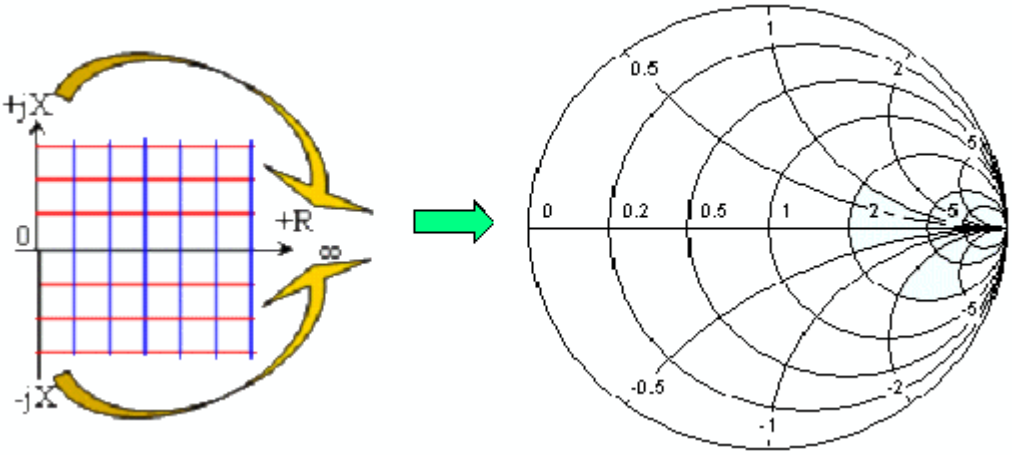
- Points with the same reactance produce arcs.

The following example shows a Smith chart with a marker used to display the stimulus value, the complex impedance Z = R + j X and the equivalent inductance L.



Smith chart construction

In a Smith chart, the impedance plane is reshaped so that the area with positive resistance is mapped into a unit circle.



The basic properties of the Smith chart follow from this construction:

- The central horizontal axis corresponds to zero reactance (real impedance). The center of the diagram represents $Z/Z_0 = 1$ which is the reference impedance of the system (zero reflection). At the left and right intersection points between the horizontal axis and the outer circle, the impedance is zero (short) and infinity (open).
- The outer circle corresponds to zero resistance (purely imaginary impedance). Points outside the outer circle indicate an active component.
- The upper and lower half of the diagram correspond to positive (inductive) and negative (capacitive) reactive components of the impedance, respectively.

Example: Reflection coefficients in the Smith chart

If the measured quantity is a complex reflection coefficient Γ (e.g. S_{11}, S_{22}), then the unit Smith chart can be used to read the normalized impedance of the DUT. The coor-

dinates in the normalized impedance plane and in the reflection coefficient plane are related as follows (see also: definition of matched-circuit (converted) impedances):

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

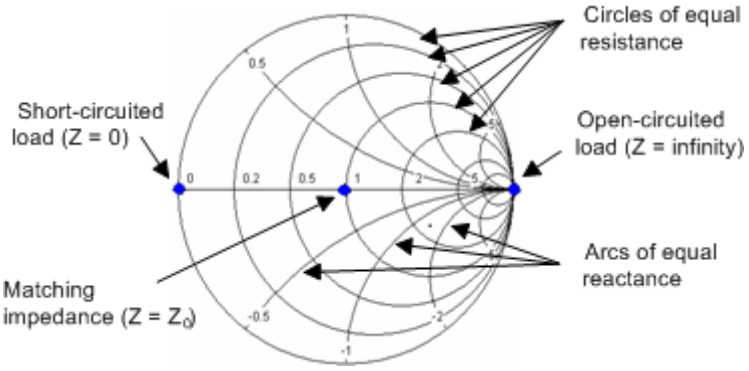
From this equation, it is easy to relate the real and imaginary components of the complex resistance to the real and imaginary parts of Γ:

$$R = \mathrm{Re}(Z/Z_0) = \frac{1-\mathrm{Re}(\Gamma)^2-\mathrm{Im}(\Gamma)^2}{[1-\mathrm{Re}(\Gamma)]^2+\mathrm{Im}(\Gamma)^2}$$

$$X = \mathrm{Im}(Z/Z_0) = \frac{2\ \mathrm{Im}(\Gamma)}{[1-\mathrm{Re}(\Gamma)]^2+\mathrm{Im}(\Gamma)^2}$$

According to the two equations above, the graphical representation in a Smith chart has the following properties:

- Real reflection coefficients are mapped to real impedances (resistances).
- The center of the Γ plane (Γ = 0) is mapped to the reference impedance Z_0, whereas the circle with |Γ| = 1 is mapped to the imaginary axis of the Z plane.
- The circles for the points of equal resistance are centered on the real axis and intersect at Z = infinity. The arcs for the points of equal reactance also belong to circles intersecting at Z = infinity (open circuit point (1, 0)), centered on a straight vertical line.



Examples for special points in the Smith chart:

- The magnitude of the reflection coefficient of an open circuit (Z = infinity, I = 0) is one, its phase is zero.
- The magnitude of the reflection coefficient of a short circuit (Z = 0, U = 0) is one, its phase is –180 deg.

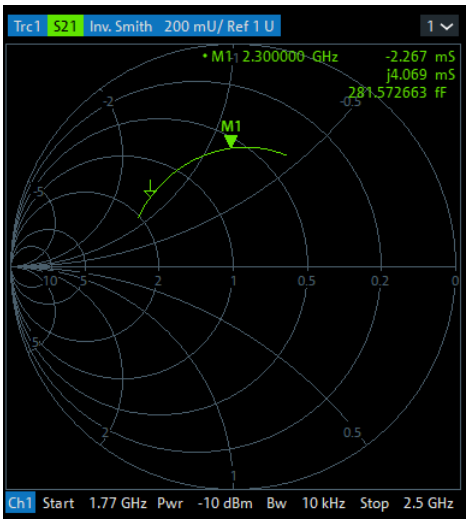
Inv Smith

For "Inv Smith" formatted traces, the response values are interpreted as complex reflection coefficients S_{ii} and represented in terms of their corresponding complex admittance $Y(S_{ii}) = G(S_{ii}) + j\ B(S_{ii})$.

In a diagram, the grid lines overlaid to a "Smith" trace correspond to points of equal conductance G and susceptance B:

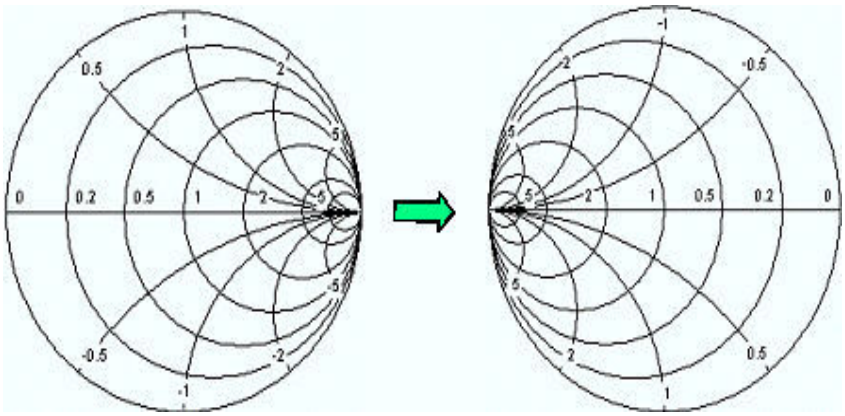
- Points with the same conductance are located on circles.
- Points with the same susceptance produce arcs.

The following example shows an inverted Smith chart with a marker used to display the stimulus value, the complex admittance Y = G + j B and the equivalent inductance L.



Inverted Smith chart construction

The inverted Smith chart is point-symmetric to the Smith chart:



The basic properties of the inverted Smith chart follow from this construction:

- The central horizontal axis corresponds to zero susceptance (real admittance). The center of the diagram represents $Y/Y_0 = 1$, where Y_0 is the reference admittance of the system (zero reflection). At the left and right intersection points between the horizontal axis and the outer circle, the admittance is infinity (short) and zero (open).
- The outer circle corresponds to zero conductance (purely imaginary admittance). Points outside the outer circle indicate an active component.

- The upper and lower half of the diagram correspond to negative (inductive) and positive (capacitive) susceptive components of the admittance, respectively.

Example: Reflection coefficients in the inverted Smith chart

If the measured quantity is a complex reflection coefficient Γ (e.g. S_{11}, S_{22}), then the inverted Smith chart can be used to read the normalized admittance of the DUT. The coordinates in the normalized admittance plane and in the reflection coefficient plane are related as follows (see also: definition of matched-circuit (converted) admittances):

$$Y/Y_0 = \frac{1-\Gamma}{1+\Gamma}$$

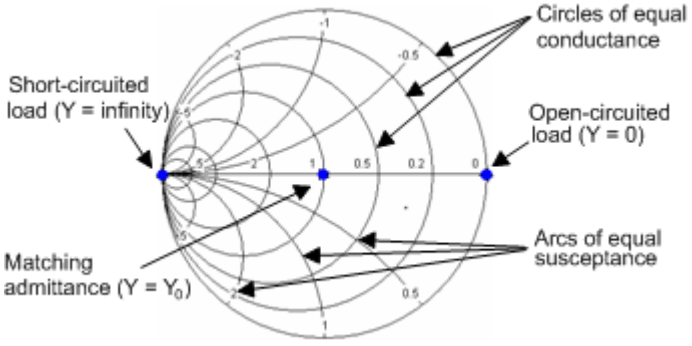
From this equation, it is easy to relate the real and imaginary components of the complex admittance to the real and imaginary parts of Γ:

$$G = \mathrm{Re}(Y/Y_0) = \frac{1-\mathrm{Re}(\Gamma)^2-\mathrm{Im}(\Gamma)^2}{[1+\mathrm{Re}(\Gamma)]^2+\mathrm{Im}(\Gamma)^2}$$

$$B = \mathrm{Im}(Y/Y_0) = \frac{-2\ \mathrm{Im}(\Gamma)}{[1+\mathrm{Re}(\Gamma)]^2+\mathrm{Im}(\Gamma)^2}$$

According to the two equations above, the graphical representation in an inverted Smith chart has the following properties:

- Real reflection coefficients are mapped to real admittances (conductances).
- The center of the Γ plane (Γ = 0) is mapped to the reference admittance Y_0, whereas the circle with |Γ| = 1 is mapped to the imaginary axis of the Y plane.
- The circles for the points of equal conductance are centered on the real axis and intersect at Y = infinity. The arcs for the points of equal susceptance also belong to circles intersecting at Y = infinity (short circuit point (–1, 0)), centered on a straight vertical line.



Examples for special points in the inverted Smith chart:

- The magnitude of the reflection coefficient of a short circuit (Y = infinity, U = 0) is one, its phase is –180 deg.
- The magnitude of the reflection coefficient of an open circuit (Y = 0, I = 0) is one, its phase is zero.

4.2.3.3 Measured quantities and trace formats

The analyzer allows any combination of a display format and a measured quantity. The following rules can help to avoid inappropriate formats and find the format that is ideally suited to the measurement task.

- All formats are suitable for the analysis of reflection coefficients S_{ii}. The formats "SWR", "Smith" and "Inv Smith" lose their original meaning (standing wave ratio, normalized impedance or admittance) if they are used for transmission S-parameters, ratios and other quantities.
- For complex "Impedances", "Admittances", "Z-parameters", and "Y-parameters" generally a Cartesian format or the polar format is suitable.
- For the real valued Stability factors and DC measurement results, one of the Cartesian formats "Lin Mag" or "Real" should be used. In complex formats, real numbers represent complex numbers with zero imaginary part.

The following table gives an overview of recommended display formats.

	Complex dimensionless quantities: **S-parameters and ratios**	**Complex quantities with dimensions:** **Wave quantities, Z-parameters, Y-parameters, impedances, admittances**	**Real quantities:** **Stability Factors, DC 1 ... 4, PAE**
Lin Mag	ON	ON (default for Z-parameters, Y-parameters, impedances, admittances)	ON (default)
dB Mag	ON (default)	ON (default for wave quantities)	–
Phase	ON	ON	–
Real	ON	ON	ON
Imag	ON	ON	–
Unwrapped Phase	ON	ON	–
Smith	ON (reflection coefficients S_{ii})	–	–
Polar	ON	–	–
Inverted Smith	ON (reflection coefficients S_{ii})	–	–
SWR	ON (reflection coefficients S_{ii})	–	–
Delay	ON (transmission coefficients S_{ij})	–	–

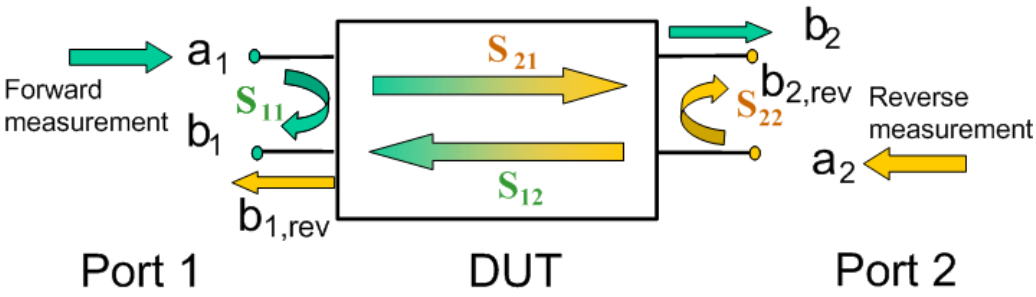
The default formats are activated automatically when the measured quantity is changed.

4.3 Measurement results

This section gives an overview of the measurement results of the network analyzer and the meaning of the different measured quantities. All quantities can be selected in the "Meas" softtool (function key Trace – [Meas]).

4.3.1 S-parameters

S-parameters are the basic measured quantities of a network analyzer. They describe how the DUT modifies a signal that is transmitted or reflected in forward or reverse direction. For a 2-port measurement, the signal flow is as follows.



The figure above is sufficient for the definition of S-parameters but does not necessarily show the complete signal flow. In fact, if the source and load ports are not ideally matched, part of the transmitted waves are reflected off the receiver ports. An additional a_2 contribution occurs in forward measurements, and an a_1 contribution occurs in reverse measurements. The 7-term calibration types Txx take these additional contributions into account.

The scattering matrix links the incident waves a_1, a_2 to the outgoing waves b_1, b_2 according to the following linear equation:

$$\begin{bmatrix} b_1 \\ b_2 \end{bmatrix} = \begin{bmatrix} S_{11} & S_{12} \\ S_{21} & S_{22} \end{bmatrix} \begin{bmatrix} a_1 \\ a_2 \end{bmatrix}$$

Meaning of 2-port S-parameters

The four 2-port S-parameters can be interpreted as follows:

- S_{11} is the reflection coefficient of DUT port 1, i.e. the ratio between outgoing wave b_1 and incident wave a_1 in a forward measurement with matched DUT port 2:
  $S_{11} = b_1 / a_1$, if $|a_1| > 0$ and $a_2 = 0$
- S_{21} is the forward transmission coefficient, defined as the ratio between outgoing wave $b_2$and incident wave a_1 in a forward measurement with matched DUT port 2:
  $S_{21} = b_2 / a_1$, if $|a_1| > 0$ and $a_2 = 0$
- S_{12} is the reverse transmission coefficient, defined as the ratio between outgoing wave $b_1$and incident wave a_2 in a forward measurement with matched DUT port 1:
  $S_{12} = b_1 / a_2$, if $|a_2| > 0$ and $a_1 = 0$
- S_{22} is the reflection coefficient of port 2, i.e. the ratio between outgoing wave b_2 and incident wave a_2 in a forward measurement with matched DUT port 1:
  $S_{22} = b_2 / a_2$, if $|a_2| > 0$ and $a_1 = 0$

Meaning of squared amplitudes

The squared amplitudes of the incident and outgoing waves and of the matrix elements have a simple meaning:

Table 4-3: Squared S-parameters

$\|a_i\|^2$	Available incident power (= the power provided by a generator with a source impedance equal to the reference impedance Z_0) at DUT port i=1,2
$\|b_i\|^2$	Reflected power at DUT port i=1,2
$10 \log\|S_{ii}\|^2 = 20 \log\|S_{ii}\|$	Reflection loss at DUT port i=1,2
$10 \log\|S_{21}\|^2 = 20 \log\|S_{21}\|$	Insertion loss of forward transmission
$10 \log\|S_{12}\|^2 = 20 \log\|S_{12}\|$	Insertion loss of reverse transmission

4.3.1.1 Multiport S-parameters

The multiport S-parameters extend the standard 2-port S-parameters to a larger number of incident and outgoing waves. For a 4-port DUT,

$$\begin{bmatrix} b_1 \\ b_2 \\ b_3 \\ b_4 \end{bmatrix} = \begin{bmatrix} S_{11} & S_{12} & S_{13} & S_{14} \\ S_{21} & S_{22} & S_{23} & S_{24} \\ S_{31} & S_{32} & S_{33} & S_{34} \\ S_{41} & S_{42} & S_{43} & S_{44} \end{bmatrix} * \begin{bmatrix} a_1 \\ a_2 \\ a_3 \\ a_4 \end{bmatrix}$$

where again a_i (i = 1 to 4) denote the incident, b_i (i = 1 to 4) denote the outgoing waves, and the S-parameters are expressed as S_{ij} (i,j = 1 to 4).

The indices of the S-parameters described so far number the output and input ports of a DUT; the parameters are referred to as single-ended S-parameters. The S-parameter description can also be used to differentiate between different propagation modes of the waves at the output and input ports. This results in the so-called mixed mode S-parameters. The analyzer measures either single-ended or mixed mode S-parameters.

4.3.1.2 Redefined S-parameters

The analyzer firmware allows you to redefine the physical VNA ports, simply by specifying:

- a reference receiver ("a wave")
- a measurement receiver ("b wave")
- a generator ("Source" VNA or external generator port)

As a consequence, the measured S-parameters and other measured quantities are also redefined.

- The receivers and generators can be freely assigned, but without reusing the same (original) physical port in different (redefined) ports. See Section 5.17.8.2, "Define Physical Ports dialog", on page 864.
- Redefined physical ports are global, persistent settings, i.e. they are valid for all recall sets. A [Preset] does not reset the physical port configuration.
- Redefining physical ports causes a factory reset.
  The factory reset deletes all switch matrix RF connections.

Redefining physical ports can be used to insert external components (e.g. external signal separating devices, power amplifiers etc.) into the signal path. This allows you to develop custom measurements, e.g. to test high-power devices and extend the dynamic range.

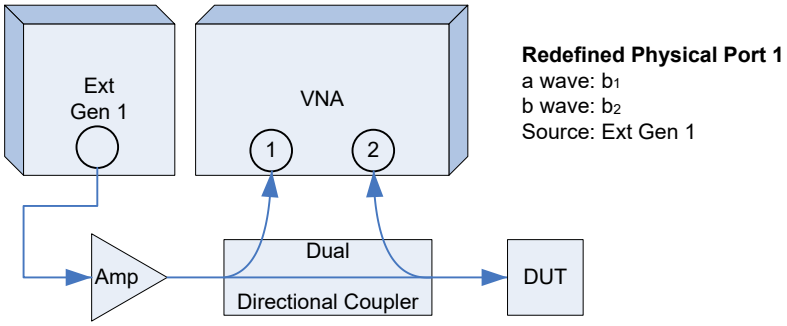


Figure 4-6: Redefined physical port 1 (port 2 not used)

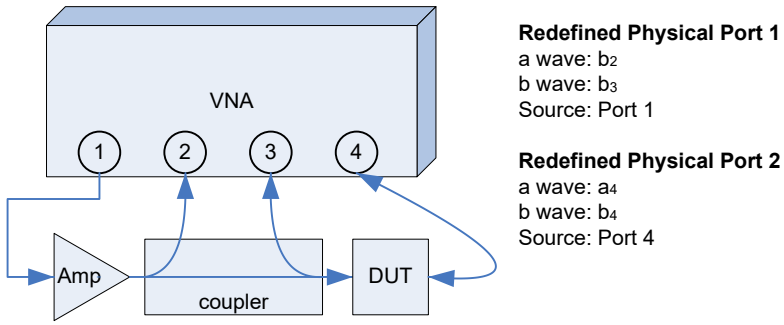


Figure 4-7: Two redefined ports

NOTICE

High Signal Power

When dealing with external signal amplification, make sure that:

- the signals fed to the analyzer are within the allowed range
- during calibration the calibration standards meet the requirements in terms of their power handling capacity

Attenuator pads can be used to adapt the power levels.

4.3.2 Reference impedances

Changing the reference impedances of the analyzer ports is often referred to as renormalization of port impedances. Renormalization means that the measurement results measured at 50 Ω (75 Ω) are converted into results at arbitrary port impedance.

- Renormalization of the physical port impedances affects, e.g., S-parameters and wave quantities in "Power" representation.

- Renormalization of the balanced port impedances affects all measured quantities that the analyzer provides for balanced ports.

The default reference impedance of a physical port is equal to the reference impedance of the connector type assigned to the port (50 Ω or 75 Ω). It can be defined as a complex value. For balanced ports, it is possible to define separate complex reference impedances for differential and for common mode.

The default values for the balanced port reference impedances are derived from the default reference impedance of the physical analyzer ports (Z_0 = 50 Ω):

- The default value for the differential mode is Z_{0d} = 100 Ω = 2*Z_0.
- The default value for the common mode is Z_{0c} = 25 Ω = Z_0/2

Renormalization can be based on two alternative waveguide circuit theories whose conversion formulas may yield different results if the reference impedance of at least one test port has a non-zero imaginary part.

Conversion formula for wave quantities and S-parameters

Renormalization transforms the "raw" S-matrix S_0 for the default reference impedances Z_{0i} (with physical port number index i = 1,2,...,n) into a "renormalized" S-matrix S_1 for the modified reference impedances Z_{1i}. In terms of raw and renormalized wave quantities a_{0i}, b_{0i} and a_{1i}, b_{1i}, S_0 and S_1 are defined as follows:

$$\begin{pmatrix} b_{01} \\ b_{02} \\ \dots \\ b_{0n} \end{pmatrix} = S_0 \cdot \begin{pmatrix} a_{01} \\ a_{02} \\ \dots \\ a_{0n} \end{pmatrix}; \quad \begin{pmatrix} b_{11} \\ b_{12} \\ \dots \\ b_{1n} \end{pmatrix} = S_1 \cdot \begin{pmatrix} a_{11} \\ a_{12} \\ \dots \\ a_{1n} \end{pmatrix}.$$

The renormalized wave quantities (a_1 and b_1) and the S-matrix S_1 can be calculated from S_0 and the reference impedances Z_{0i}, Z_{1i} according to two alternative waveguide circuit theories.

1. **Traveling waves**
   In the model of Marks and Williams ("A General Waveguide Circuit Theory"), the wave quantities a and b are transformed as follows:

$$\begin{pmatrix} a_{1i} \\ b_{1i} \end{pmatrix} = \frac{1}{2Z_{0i}} \left| \frac{Z_{0i}}{Z_{1i}} \right| \sqrt{\frac{\mathrm{Re}(Z_{1i})}{\mathrm{Re}(Z_{0i})}} \cdot \begin{pmatrix} Z_{0i} + Z_{1i} & Z_{0i} - Z_{1i} \\ Z_{0i} - Z_{1i} & Z_{0i} + Z_{1i} \end{pmatrix} \cdot \begin{pmatrix} a_{0i} \\ b_{0i} \end{pmatrix}$$

   The renormalized S-matrix S1 is calculated as:

$$S_1 = P^{-1}(S_0 - \gamma)(E - \gamma S_0)^{-1} P$$

   with the unit matrix E and two additional matrices with the elements

$$\gamma_{ii} = \frac{Z_{1i} - Z_{0i}}{Z_{1i} + Z_{0i}}$$

$$P_{ii} = \frac{Z_{0i}}{Z_{0i} + Z_{1i}} \left| \frac{Z_{1i}}{Z_{0i}} \right| \sqrt{\frac{\mathrm{Re}(Z_{0i})}{\mathrm{Re}(Z_{1i})}}$$

2. **Power waves**
   In the model of Kurokawa ("Power Waves and the Scattering Matrix"), the wave quantities a and b are transformed as follows:

$$\begin{pmatrix} a_{1i} \\ b_{1i} \end{pmatrix} = \frac{1}{2\sqrt{\mathrm{Re}(Z_{0i})\,\mathrm{Re}(Z_{1i})}} \cdot \begin{pmatrix} \overline{Z_{0i}} + Z_{1i} & Z_{0i} - Z_{1i} \\ \overline{Z_{0i}} - \overline{Z_{1i}} & Z_{0i} + \overline{Z_{1i}} \end{pmatrix} \cdot \begin{pmatrix} a_{0i} \\ b_{0i} \end{pmatrix}$$

   The renormalized S-matrix S1 is calculated as:

$$S_1 = A^{-1}\left(S_0 - \overline{\Gamma}\right)\left(E - \Gamma S_0\right)^{-1} \overline{A}$$

   with the unit matrix E and two additional matrices with the elements

$$\Gamma_{ii} = \frac{Z_{1i} - Z_{0i}}{Z_{1i} + \overline{Z_{0i}}}$$

$$A_{ii} = \frac{1 - \overline{\Gamma_{ii}}}{\left|1 - \Gamma_{ii}\right|} \sqrt{\left|1 - \Gamma_{ii}\overline{\Gamma_{ii}}\right|}$$

4.3.3 Impedance parameters

An impedance is the complex ratio between a voltage and a current. The analyzer provides two independent sets of impedance parameters:

- Converted impedances (each impedance parameter is obtained from a single S-parameter)
- Z-parameters (complete description of an n-port DUT)

4.3.3.1 Converted impedances

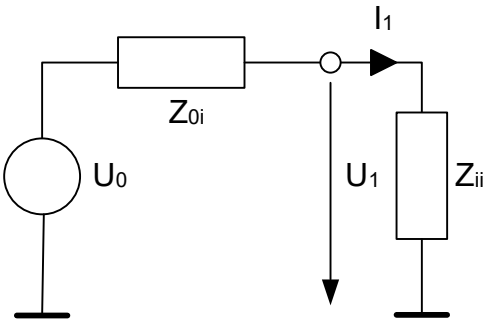
The converted impedance parameters Z_{ij} ($1 \le i,j \le n$) describe the input impedances of an n-port DUT with fully matched outputs. The analyzer converts a single measured S-parameter S_{ij} to determine the corresponding converted impedance, under the assumption that each of the other ports is terminated with its respective reference impedance Z_{0k} (matched-circuit parameters). As a result, converted impedances cannot completely describe general n-port DUTs.

The extension of the impedances to more ports and mixed mode measurements is analogous to S-parameters. Z_{dd44} is the differential mode input impedance at port 4 of a DUT that is terminated at its other ports with the reference impedance Z_0.

The Converted admittances are defined as the inverse of the converted impedances.

Reflection impedance

The converted impedance Z_{ii} ($1 \leq i \leq n$) describes the input impedance at port i of the DUT.



Example:

For a 2-port DUT that is terminated at its output with the reference impedance Z_{02}, Z_{11} is the input impedance (matched-circuit impedance measured in a forward reflection measurement).

A converted impedance Z_{11} completely describes a one-port DUT.

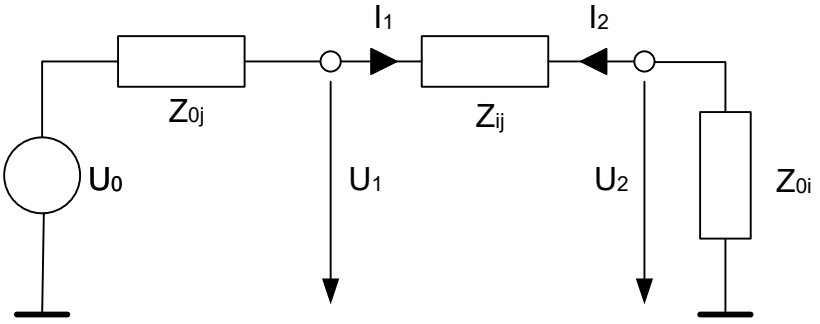
The calculation formula of the converted reflection impedances Z_{ii} depends on the waveguide circuit theory according to which Reference impedances are calculated.

Table 4-4: Calculation of converted reflection impedances

Traveling Waves	Power Waves
$Z_{ii} = Z_{0i}\frac{1+S_{ii}}{1-S_{ii}}$	$Z_{ii} = \frac{\overline{Z_{0i}} + S_{ii}Z_{0i}}{1-S_{ii}}$

Series transmission impedance

A two-port transmission parameter Z_{ij} ($i \neq j$) can describe a pure serial impedance between the two ports.



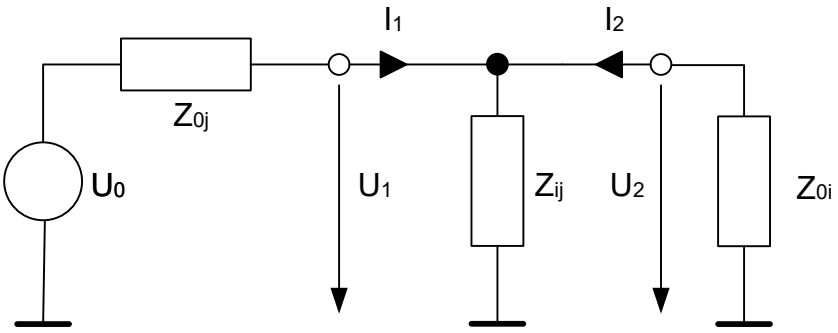
The calculation formula of a converted serial transmission impedance Z_{ij} depends on the waveguide circuit theory according to which Reference impedances are calculated.

Table 4-5: Calculation of converted series transmission Impedances

Traveling Waves	Power Waves
$Z = 2e^{j\varphi(Z_{0i})}\sqrt{\left\lvert\frac{\mathrm{Re}(Z_{0i})}{\mathrm{Re}(Z_{0j})}\right\rvert\frac{\lvert Z_{0j}\rvert}{S_{ij}}} - (Z_{0i} + Z_{0j})$	$Z = 2\,\mathrm{sgn}(\mathrm{Re}(Z_{0i}))\frac{\sqrt{\lvert\mathrm{Re}(Z_{0i})\mathrm{Re}(Z_{0j})\rvert}}{S_{ij}} - (Z_{0i} + Z_{0j})$

Parallel transmission impedance

A two-port transmission parameter Z_{ij} ($i \neq j$) can also describe a parallel impedance between the two ports.



The calculation formula of a converted parallel transmission impedance Z_{ij} depends on the waveguide circuit theory according to which Reference impedances are calculated.

Table 4-6: Calculation of Converted Parallel Transmission Impedances

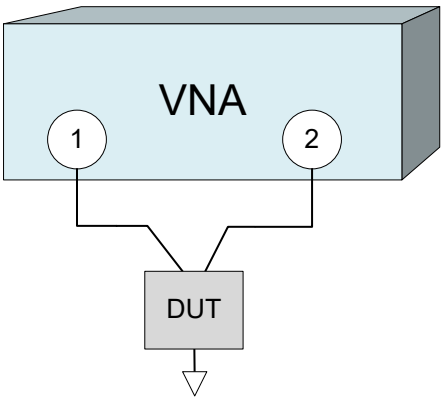
Traveling Waves	Power Waves
$Z = \frac{Z_{0i}Z_{0j}S_{ij}}{2\left(Z_{0i}\left\lvert\frac{Z_{0j}}{Z_{0i}}\right\rvert\sqrt{\left\lvert\frac{\mathrm{Re}(Z_{0i})}{\mathrm{Re}(Z_{0j})}\right\rvert} - S_{ij}\frac{Z_{0i}+Z_{0j}}{2}\right)}$	$Z = \frac{Z_{0j}S_{ij}}{\sqrt{\left\lvert\frac{\mathrm{Re}(Z_{0j})}{\mathrm{Re}(Z_{0i})}\right\rvert}\left(1 + e^{-2\varphi(Z_{0i})}\right) - S_{ij}\left(1 + \frac{Z_{0j}}{Z_{0i}}\right)}$

Shunt-thru measurements

The shunt-thru method is used for measuring very low impedances. A typical application is measurements on power distribution network (PDN) components, such as bypass capacitors and DC/DC converters.

The R&S ZNB uses S_{ij} ($i \neq j$) to calculate the DUT impedance using the formula:

$$Z = \frac{50\,\Omega}{2} \cdot \frac{S_{ij}}{1 - S_{ij}}$$



4.3.3.2 Z-parameters

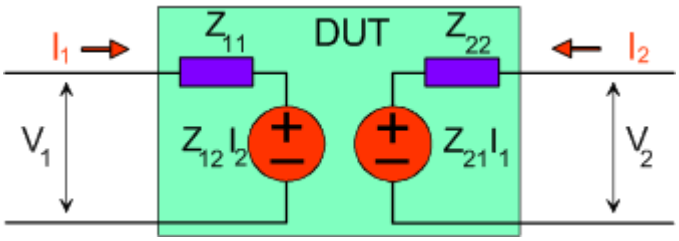
The Z-parameters describe the impedances of a DUT with open output ports (impedance = 0). The analyzer provides the full set of Z-parameters including the transfer impedances (i.e. the complete nxn Z-matrix for an n-port DUT).

This means that Z-parameters can be used as an alternative to S-parameters (or Y-parameters) to characterize a linear n-port network completely.

2-port Z-parameters

In analogy to S-parameters, Z-parameters are expressed as Z_{ij}, where i denotes the measured and j the stimulated port.

The Z-parameters for a two-port are based on a circuit model that can be expressed with two linear equations:



$$V_1 = Z_{11}I_1 + Z_{12}I_2$$
$$V_2 = Z_{21}I_1 + Z_{22}I_2$$

Meaning of Z-parameters

The four 2-port Z-parameters can be interpreted as follows:

- Z_{11} is the input impedance, defined as the ratio of the voltage V_1 to the current I_1, measured at port 1 (forward measurement with open output, $I_2 = 0$).
- Z_{21} is the forward transfer impedance, defined as the ratio of the voltage V_2 to the current I_1 (forward measurement with open output, $I_2 = 0$).
- Z_{12} is the reverse transfer impedance, defined as the ratio of the voltage V_1 to the current I_2 (reverse measurement with open input, $I_1 = 0$).

- Z_{22} is the output impedance, defined as the ratio of the voltage V_2 to the current I_2, measured at port 2 (reverse measurement with open input, $I_1 = 0$).

Z-parameters can be easily extended to describe circuits with more than two ports or several modes of propagation.

4.3.4 Admittance parameters

An admittance is the complex ratio between a current and a voltage. The analyzer provides two independent sets of admittance parameters:

- Converted admittances (each admittance parameter is obtained from a single S-parameter)
- Y-parameters (complete description of the n-port DUT)

4.3.4.1 Converted admittances

The converted admittance parameters describe the input admittances of a DUT with fully matched outputs. The converted admittances are the inverse of the Converted impedances.

4.3.4.2 Y-parameters

The Y-parameters describe the admittances of a DUT with output ports terminated in a short circuit (voltage = 0). The analyzer provides the full set of Y-parameters including the transfer admittances (i.e. the complete n x n Y-matrix for an n-port DUT).

This means that Y-parameters can be used as an alternative to S-parameters (or Z-parameters) to characterize a linear n-port network completely.

2-port Y-parameters

In analogy to S-parameters, Y-parameters are expressed as $Y_{<out><in>}$, where <out> and <in> denote the output and input port numbers of the DUT. In analogy to Z-parameters, the Y-parameters for a two-port are based on a circuit model that can be expressed with two linear equations:

$$I_1 = Y_{11}V_1 + Y_{12}V_2$$
$$I_2 = Y_{21}V_1 + Y_{22}V_2$$

Meaning of Y-parameters

The four 2-port Y-parameters can be interpreted as follows:

- Y_{11} is the input admittance, defined as the ratio of the current I_1 to the voltage V_1, measured at port 1 (forward measurement with output terminated in a short circuit, $V_2 = 0$).
- Y_{21} is the forward transfer admittance, defined as the ratio of the current I_2 to the voltage V_1 (forward measurement with output terminated in a short circuit, $V_2 = 0$).

- Y_{12} is the reverse transfer admittance, defined as the ratio of the current I_1 to the voltage V_2 (reverse measurement with input terminated in a short circuit, $V_1 = 0$).
- Y_{22} is the output admittance, defined as the ratio of the current I_2 to the voltage V_2, measured at port 2 (reverse measurement with input terminated in a short circuit, $V_1 = 0$).

Y-parameters can be easily extended to describe circuits with more than two ports or several modes of propagation.

4.3.5 Wave quantities and ratios

The elements of the S-, Z- and Y-matrices represent fixed ratios of complex wave amplitudes. As long as the assumption of linearity holds, the S-, Z- and Y-parameters are independent of the source power.

The network analyzer provides two additional sets of measurement parameters which have an unambiguous meaning even if the DUT is measured outside its linear range:

- *Wave quantities* provide the power of any of the transmitted or received waves.
- *Ratios* provide the complex ratio of any combination of transmitted or received wave quantities.

In contrast to S-, Z- and Y-parameters, wave quantities and ratios are not system-error corrected.

To increase the accuracy or to correct a possible attenuation in the source signal path, it is recommended to perform a power calibration (see Section 4.5.6, "Scalar power calibration", on page 187).

4.3.5.1 Wave quantities

A wave quantity measurement provides the power of any of the transmitted or received waves. The power can be displayed in voltage units (e.g. V or dBmV) or equivalent power units (e.g. W or dBm).

Examples for using wave quantities

The wave quantities provide the power at the different receive ports of the analyzer. This is different from an S-parameter measurement, where the absolute power of a linear device is canceled. Wave quantities are therefore suitable for the following measurement tasks:

- Analysis of nonlinearities of the DUT.
- Use of the analyzer as a selective power meter.
  To increase the accuracy or to correct a possible attenuation in the source signal path, it is recommended to perform a power calibration (see Section 4.5.6, "Scalar power calibration", on page 187).

The notation for wave quantities is as follows:

- "a_i Src Port j" denotes the wave incoming at DUT port i, when DUT port j is stimulated.
  a_i is detected at the reference receiver of the VNA port connected to DUT port i.
- "b_i Src Port j" denotes the wave outgoing at DUT port i, when DUT port j is stimulated.
  b_i is detected at the measurement receiver of the VNA port connected to DUT port i.

In a standard forward S-parameter measurement, a_1 Src Port 1 is the incident wave and b_1 Src Port 1 is the reflected wave at DUT port 1.

4.3.5.2 Ratios

A ratio measurement provides the complex ratio of any combination of transmitted or received wave amplitudes. Ratios complement the S-parameter measurements, where only ratios of the form b_i/a_j (ratios between outgoing and incoming waves at the DUT ports) are considered.

Examples for using ratios

A measurement of ratios is particularly suitable for the following test scenarios:

- The test setup or some of its components (e.g. active components or non-reciprocal devices) do not allow a system error correction so that a complete S-parameter measurement is not possible.
- The test setup contains frequency-converting components so that the transmitted and the received waves are at different frequencies.
- A ratio of two arbitrary waves that is not an element of the S-matrix (e.g. a ratio of the form a_i/a_j) is needed.

The notation for ratios is similar to the notation for wave quantities (see Section 4.3.5.1, "Wave quantities", on page 126). Given a source port k, any ratio between wave quantities "a_i Src Port k" and "b_j Src Port k" can be measured.

Examples:

- "b_2/a_1 Src Port 1" is the ratio of the outgoing wave at DUT port 2 and the incident wave at DUT port 1 (i.e. DUT port 1 ist stimulated). This corresponds to the forward transmission coefficient S_{21}.
- "b_1/a_1 Src Port 1" is the ratio of the outgoing wave at DUT port 1 and the incident wave at DUT port 1 (i.e. DUT port 1 ist stimulated). This corresponds to the forward reflection coefficient S_{11}.

4.3.5.3 Detector settings

The "Detector" settings select the algorithm that is used to calculate the displayed measurement points from the raw data. The "Detector" can be selected in the "More Wave Quantities" and More Ratios dialogs.

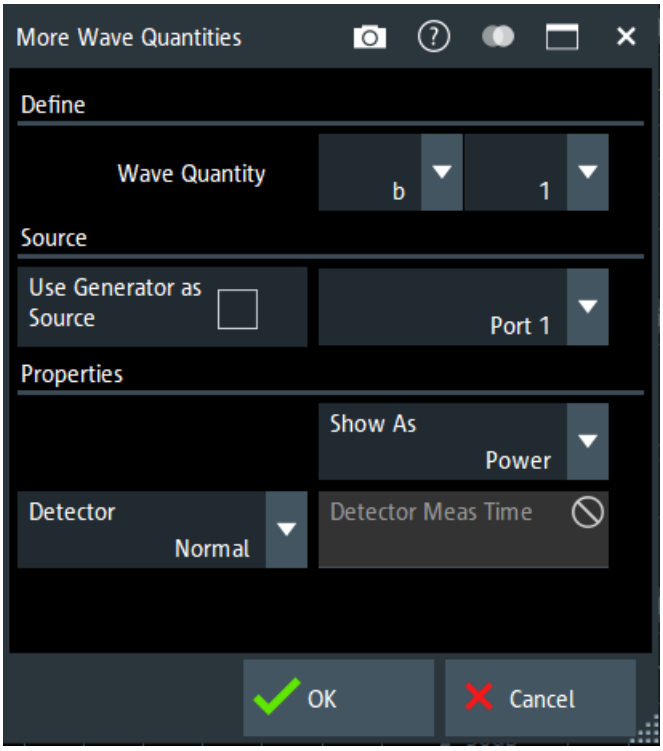


Figure 4-8: More Wave Quantities dialog

The following detectors are available:

- **"Normal"** selects the default detector mode where each valid measurement point is displayed without modification. The analyzer then proceeds to the next sweep point. Normal detector mode ensures that the measurement is performed at maximum speed and that a meaningful complex result is obtained.
- **"AVG Real Imag"** collects all valid results at each sweep point during the "Detector Meas Time" set in the "More Wave Quantities" or More Ratios dialog and calculates the complex arithmetic mean of their real and imaginary parts.
  The AVG detector is also used for gain measurements in noise figure channels.
- **AVG Mag Phase** collects all valid results at each sweep point during the "Detector Meas Time" set in the "More Wave Quantities" or More Ratios dialog. It calculates the arithmetic mean of the magnitude and phase values of these results separately.
- **RMS** collects all valid results at each sweep point during the "Detector Meas Time" set in the "More Wave Quantities" or More Ratios dialog and calculates the root mean square (RMS) of the linear magnitude of these results. Note that the phase is not evaluated in this process so that complex conversions (e. g. the calculation of real and imaginary values) do not really make sense.
  The RMS detector is also used for Noise figure measurement.

Combining different detectors

The detector setting in the More Ratios menu applies to both the numerator and the denominator wave quantity. To allow for different detector settings, measure the numerator and denominator wave quantities individually and use trace functions to calculate the ratio. A possible application is the comparison of different detector settings for a particular trace.

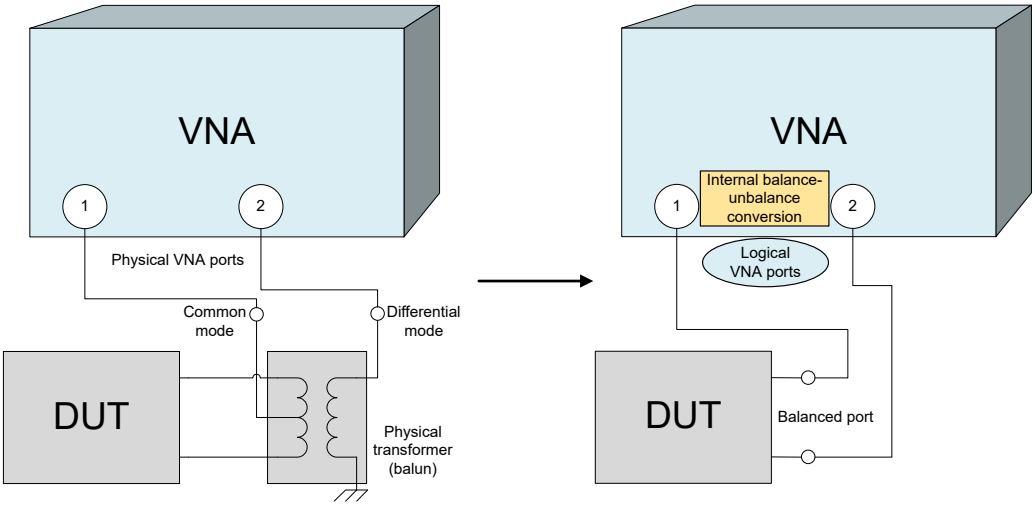
Error Messages

The analyzer generates a warning if the selected measurement time for the "AVG..." detectors is too long. At the same time, bit no. 15 in the `...INTegrity:HARDware` status register is set. Reduce the measurement time and/or reduce the IF bandwidth until the warning disappears. A warning also appears if the measurement time for the "AVG..." detectors is too short. Increase the measurement time and/or increase the IF bandwidth until the warning disappears.

4.3.6 Unbalance-balance conversion

Unbalance-balance conversion is the simulation of one or more unbalance-balance transformers (baluns) integrated in the measurement circuit. It converts the DUT ports from an unbalanced state into a balanced state and virtually separates the differential and common mode signals. The analyzer measures the unbalanced state but converts the results and calculates mixed mode parameters, e.g. mixed mode S-parameters. No physical transformer is needed.

To perform balanced measurements, a pair of physical analyzer ports is combined to form a logical port. The balanced port of the DUT is directly connected to the analyzer ports. For a two-port analyzer, a single balanced port can be defined.



Unbalance-balance conversion avoids the disadvantages of real transformers:

- There is no need to fabricate test fixtures with integrated baluns for each type of DUT.

- The measurement is not impaired by the non-ideal characteristics of the balun (e.g. error tolerances, limited frequency range).
- Calibration can be performed at the DUT's ports. If necessary (e.g. to compensate for the effect of a test fixture), it is possible to shift the calibration plane using length offset parameters.
- Differential and common mode parameters can be evaluated with a single test setup.

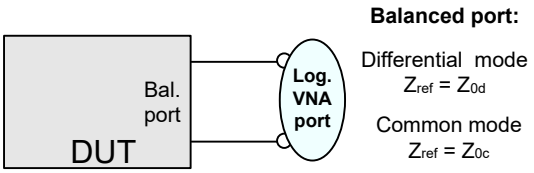
4.3.6.1 Balanced port configurations

Defining a balanced logical port requires two physical ports.

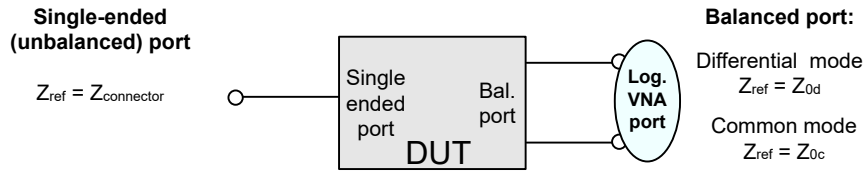
The physical ports (on the VNA and connected External switch matrices) are equivalent and can be freely combined to balanced (logical) ports. Moreover, it is possible to assign arbitrary, independent reference impedance values to each unbalanced port and to the differential and common mode of each logical port.

Example:

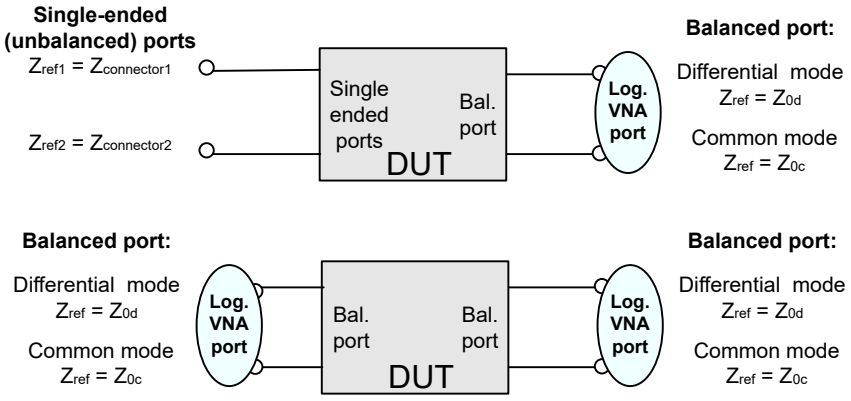
2 physical ports: Reflection measurements on 1 balanced port



3 physical ports: Reflection and transmission measurements on 1 balanced port



4 physical ports: Reflection and transmission measurements on 1 or 2 balanced ports



A balanced port configuration is defined in two steps: First, select the pairs of physical ports that you want to combine to form balanced ports. Second, define the two refer-

ence impedances for the differential and common mode at each balanced port. Both steps can be done in a single "Balanced Ports" dialog. The most commonly used balanced port configurations and impedances are predefined and can be selected in the "S-Parameter Wizard".

Depending on the test setup, the analyzer provides different types of mixed mode parameters; refer to the following sections for details.

4.3.6.2 Mixed-mode parameters

Mixed mode parameters are an extension of normal mode parameters (e.g. S-parameters, impedances and admittances) for balanced measurements. The analyzer can measure mixed mode parameters once a balanced port configuration is selected.

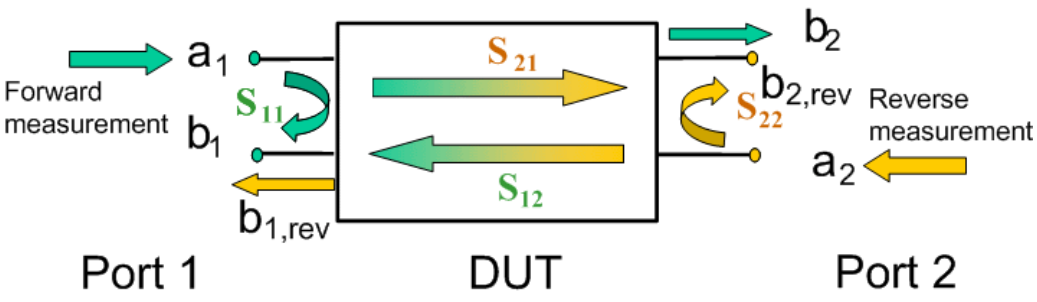
Mixed mode parameters are used to distinguish the following three port modes:

- s: Single-ended (for unbalanced ports)
- d: Differential mode (for balanced ports)
- c: Common mode (for balanced ports)

The notation of a general S-parameter is $S_{<mout><min><out><in>}$, where <mout> and <min> denote the output and input port modes, <out> and <in> denote the output and input port numbers.

Meaning of 2-port mixed mode S-parameters

The mixed mode 2-port S-parameters can be interpreted as follows:



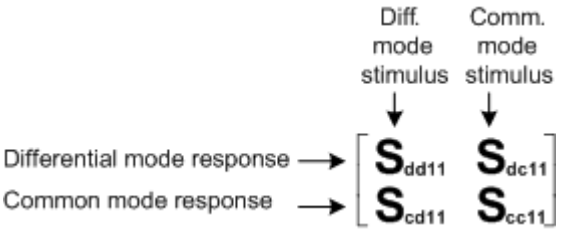
- $S_{<mout><min>11}$ is the mixed mode input reflection coefficient, defined as the ratio of the wave quantities b_1 (mode mout) to a_1 (mode min), measured at PORT 1 (forward measurement with matched output and $a_2 = 0$).
- $S_{<mout><min>21}$ is the mixed mode forward transmission coefficient, defined as the ratio of the wave quantities b_2 (mode mout) to a_1 (mode min) (forward measurement with matched output and $a_2 = 0$).
- $S_{<mout><min>12}$ is the mixed mode reverse transmission coefficient, defined as the ratio of the wave quantities b_1 (mode mout) (reverse measurement with matched input, b_1' in the figure above and $a_1 = 0$) to a_2 (mode min).
- $S_{<mout><min>22}$ is the mixed mode output reflection coefficient, defined as the ratio of the wave quantities b_2 (mode mout) (reverse measurement with matched input, b_2' in the figure above and $a_1 = 0$) to a_2 (mode min), measured at PORT 2.

If <mout> is different from <min>, the S-parameters are called mode conversion factors.

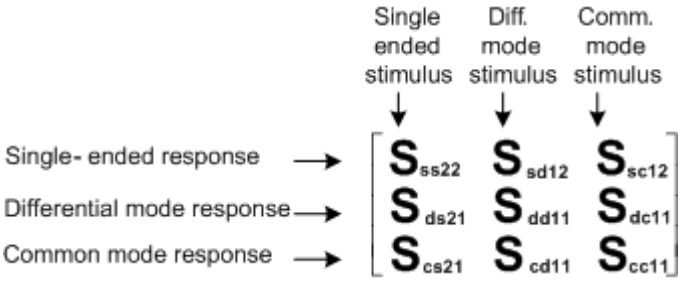
Mixed-mode parameters for different test setups

Which types of mixed mode parameter are available depends on the measured device and the port configuration of the analyzer. With 4 physical test ports (on the VNA and connected External switch matrices), the following examples of mixed mode parameters can be measured.

1. DUT with only single-ended ports: No balanced port definition necessary, the analyzer provides single-ended multiport parameters.
2. DUT with one balanced port: Only reflection and mode conversion measurements with differential and common mode parameters.



3. DUT with one balanced and one single-ended port.



4. DUT with two balanced ports or one balanced and two single-ended ports. Both device types are fully characterized by 4x4 mixed mode S-matrices.

4.3.6.3 Imbalance and common mode rejection

An ideal unbalance-balance transformer (balun) converts an unbalanced signal into a balanced one and vice versa. When it is driven with an unbalanced signal at its physical port 1 (= single-ended logical port 1), unbalanced signals with equal amplitude and opposite phase appear at physical ports 2 and 4 (forming balanced logical port 2). This means that the ratio $-S_{21}/S_{41}$ of the physical transmission coefficients of an ideal balun equals 1. This ratio is called **imbalance**; it is a measure for the deviation of the balun from ideality. The general definition of the transmission imbalance between two different ports (at least one of them balanced) is given below.

For a DUT with two balanced ports (e.g. an amplifier), the ratio between the (wanted) differential mode power gain and the (unwanted) common mode power gain is called

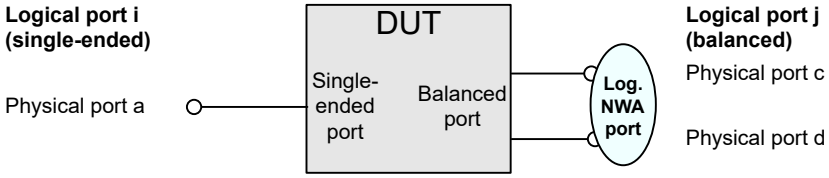
common-mode rejection ratio (CMRR). It can be calculated as $|S_{dd21}|/|S_{cc21}|$ (see Section 4.3.6.2, "Mixed-mode parameters", on page 131). The general definition of the complex CMRR between two ports (at least one of them balanced) is given below.

Imbalance and common-mode rejection ratio can only be measured if more than 2 test ports are available on the VNA and connected switch matrices.

General Definition

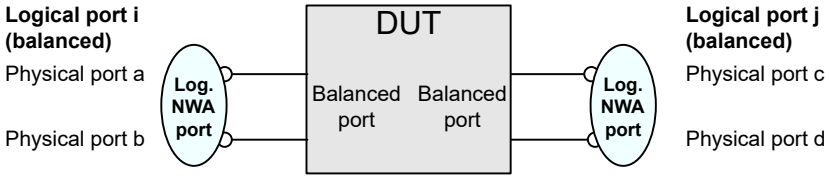
In general, imbalance and CMRR are quantities with two numeric indices, indicating the logical output port and the logical input port of the DUT during the measurement ($Imb_{<out><in>}$, $CMRR_{<out><in>}$).

- DUT with single-ended logical port i and balanced logical port j:



  - $Imb_{ji} = -S_{ca}/S_{da}$ and $Imb_{ij} = -S_{ac}/S_{ad}$
  - $CMRR_{ji} = S_{dsji}/S_{csji}$ and $CMRR_{ij} = S_{sdij}/S_{scij}$

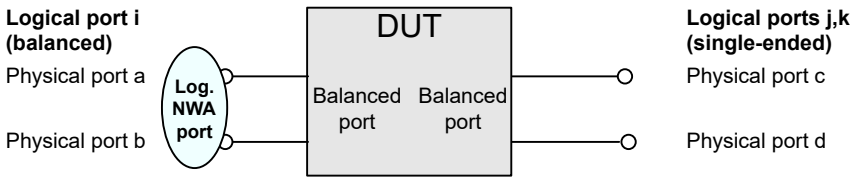
- DUT with balanced logical ports i and j:



  - $Imb_{ji} = -(S_{ca} - S_{cb})/(S_{da} - S_{db})$, for $i \neq j$
  - $CMRR_{ji}=S_{ddji}/S_{ccji}$

Differential Imbalance

A (differential) imbalance can also be calculated if the second balanced DUT port is connected to two single-ended logical ports:



$Imb_{jk-i} = -(S_{ca} - S_{da})/(S_{cb} - S_{db})$

$Imb_{i-jk} = -(S_{ac} - S_{ad})/(S_{bc} - S_{bd})$

4.3.7 Stability factors

The stability factors K, μ1 and μ2 are real functions of the (complex) S-parameters, defined as follows:

$$K := \frac{1 - |S_{11}|^2 - |S_{22}|^2 + |S_{11} \cdot S_{22} - S_{12} \cdot S_{21}|^2}{2 \cdot |S_{12} \cdot S_{21}|}$$

$$\mu_1 := \frac{1 - |S_{11}|^2}{|S_{22} - \overline{S_{11}} \cdot (S_{11} \cdot S_{22} - S_{12} \cdot S_{21})| + |S_{12} \cdot S_{21}|}$$

$$\mu_2 := \frac{1 - |S_{22}|^2}{|S_{11} - \overline{S_{22}} \cdot (S_{11} \cdot S_{22} - S_{12} \cdot S_{21})| + |S_{12} \cdot S_{21}|}$$

where $\bar{S}$ denotes the complex conjugate of S.

Stability factors are calculated as functions of the frequency or another stimulus parameter. They provide criteria for linear stability of two-ports such as amplifiers. A linear circuit is said to be unconditionally stable if no combination of passive source or load can cause the circuit to oscillate.

- The K-factor provides a necessary condition for unconditional stability: A circuit is unconditionally stable if K>1 and an additional condition is met. The additional condition can be tested with the stability factors μ_1 and μ_2.
- The μ_1 and μ_2 factors both provide a necessary and sufficient condition for unconditional stability: The conditions μ_1>1 or μ_2>1 are both equivalent to unconditional stability. This means that μ_1 and μ_2 provide direct insight into the degree of stability or potential instability of linear circuits.

References: Marion Lee Edwards and Jeffrey H. Sinsky, "A New Criterion for Linear 2-Port Stability Using a Single Geometrically Derived Parameter", IEEE Trans. MTT, vol. 40, No. 12, pp. 2303-2311, Dec. 1992.

4.3.8 Group delay

The group delay τ_g represents the propagation time of wave through a device. τ_g is a real quantity and is calculated as the negative of the derivative of its phase response. A non-dispersive DUT shows a linear phase response, which produces a constant delay (a constant ratio of phase difference to frequency difference).

The group delay is defined as:

$$\tau_g = -\frac{d\phi_{rad}}{d\omega} = -\frac{d\phi_{deg}}{360° df}$$

where

Φ_{rad}, Φ_{deg}= phase response in radians or degrees

ω = angular velocity in radians/s

f = frequency in Hz

In practice, the analyzer calculates an approximation to the derivative of the phase response, taking a small frequency interval Δf and determining the corresponding phase change ΔΦ. The group delay is computed as

$$\tau_{g,meas} = -\frac{\Delta\phi_{deg}}{360° \cdot \Delta f}$$

where ΔΦ/Δf is the slope of the regression line through the frequency points of aperture Δf.

Δf must be adjusted to the conditions of the measurement, e.g. it must be reduced if phase slope fluctuates significantly over frequency. Otherwise group delay variations are flattened out.

Note that the input value "Aperture Points" does not define the number of frequency points, but the number of frequency steps between the points. I.e. "Aperture Points" is always the number of frequency points involved minus 1. If "Aperture Points" is an odd number, the part of the aperture towards lower frequencies contains one more frequency step than the part towards larger frequencies.

If the group delay is constant over the considered frequency range (non-dispersive DUT, e.g. a cable), then τ_g and $\tau_{g,meas}$ are identical and:

$$\tau_g = \frac{d\,(360° f \cdot \Delta t)}{360° d\, f} = \Delta t = \frac{L_{mech} \cdot \sqrt{\varepsilon}}{c}$$

where Δt is the propagation time of the wave across the DUT, which often can be expressed in terms of its mechanical length L_{mech}, the permittivity ε, and the velocity of light c. The product $L_{mech} \cdot \mathrm{sqrt}(\varepsilon)$ is termed the electrical length of the DUT and is always larger or equal than the mechanical length (ε > 1 for all dielectrics and ε = 1 for the vacuum).

4.4 Operations on traces

The R&S ZNB can perform more complex operations on the measured traces. Some of the operations, e.g. the time domain transform, require additional software options; see Section 4.7, "Optional extensions and accessories", on page 216.

The R&S ZNB can also check whether the measured values comply with specified limits and export trace data, using different file formats.

4.4.1 Limit check

A limit line restricts the allowed range for some or all points of a trace, i.e. for a certain range of stimulus values. Typically, limit lines are used to check whether a DUT conforms to the rated specifications (conformance testing).

- An upper limit line defines the maximum allowed values for the related stimulus range.
- A lower limit line defines the minimum allowed values for the related stimulus range.
- A ripple limit defines the maximum difference between the largest and the smallest response value for the related stimulus range. A ripple limit test is suitable, e.g., to check whether the passband ripple of a filter is within acceptable limits, irrespective of the actual transmitted power in the passband.
- A circle limit defines the acceptable values as a circular area within a complex diagram.

A limit check compares the measurement results to the limit lines, and displays a pass/fail indication.

A TTL signal at the rear panel USER PORT (for test automation) can be generated in addition, if a limit is violated.

Upper and lower limit lines are both defined as a combination of segments with a linear or logarithmic dependence between the measured quantity and the sweep variable (stimulus variable). Similar to this segmentation, ripple limits can be defined in several ranges. The limit lines (except circle limits) can be stored to a file and recalled. Data or memory traces can be used to define the segments of an upper or lower limit line. Moreover it is possible to modify the upper and lower limit lines globally by adding an offset to the stimulus or response values.

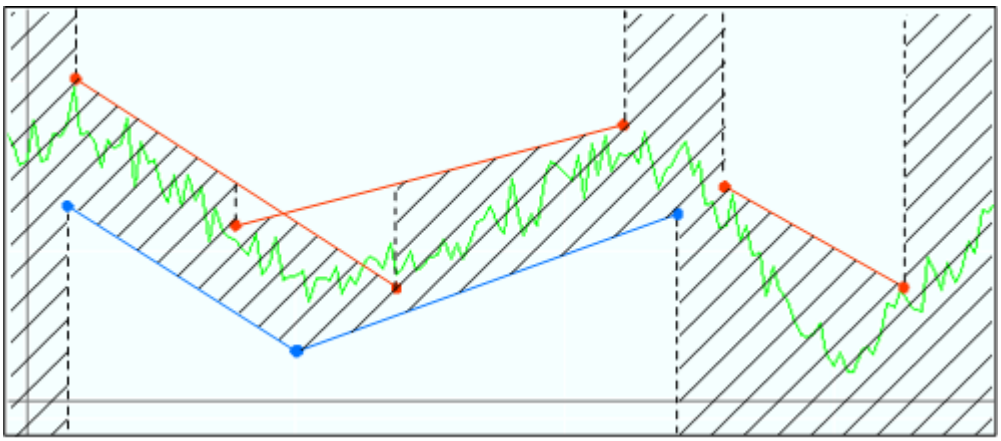
4.4.1.1 Rules for limit line definition

The analyzer places few restrictions on the definition of limit line segments.

The following rules ensure a maximum of flexibility:

- Segments do not have to be sorted in ascending or descending order (e.g. the "Start Stimulus" value of segment no. n does not have to be smaller than the "Start Stimulus" value of segment no. n+1).
- Overlapping segments are allowed. The limit check in the overlapping area is related to the tighter limit (the pass test involves a logical AND operation).
- Gaps between segments are allowed and equivalent to switching off an intermediate limit line segment.
- Limit lines can be partially or entirely outside the sweep range, however, the limits are only checked at the measurement points.

The following figure shows a limit line consisting of 3 upper and 2 lower limit line segments. To pass the limit check, the trace must be confined to the shaded area.

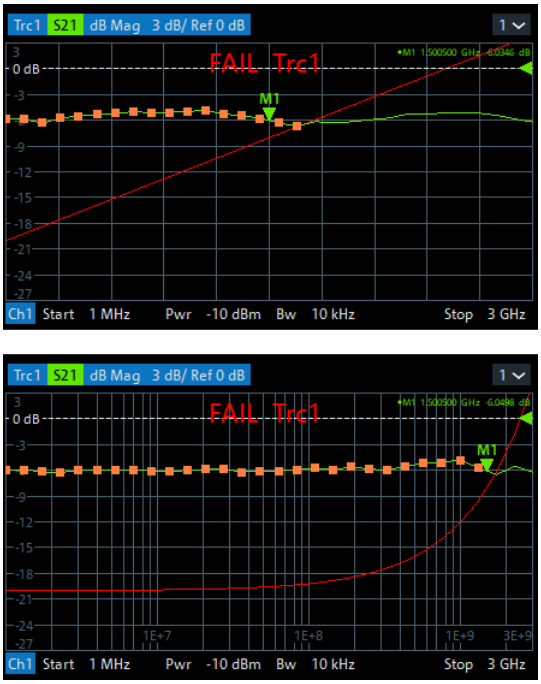


As a consequence of the limit line rules, a DUT always passes the limit check if no limit lines are defined.

Formula-defined limit lines

A limit line doesn't have to be straight. You can also use a custom formula to define it.

When the sweep axis is changed from linear frequency sweep to logarithmic sweeps, straight limit lines are transformed into exponential curves. The sweep points are redistributed along the x-axis, so the number of failed points can change.

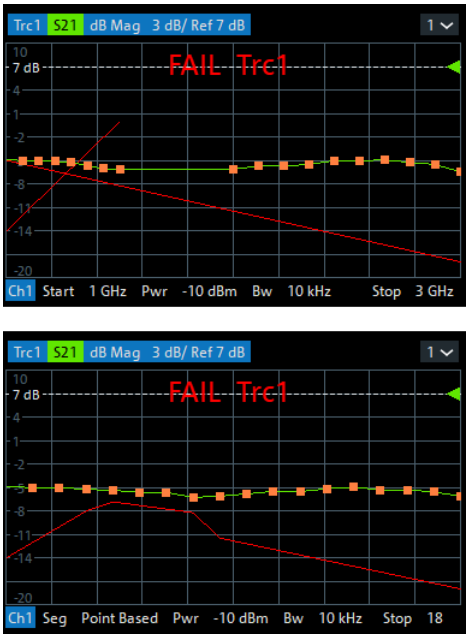


Logarithmic interpolation

The analyzer offers a logarithmic interpolation mode that allows you to carry over the limit line definition to logarithmic sweeps.

While "Show Limit Line" is active, the diagrams display all limit line segments.

Exception: In a segmented frequency sweep with point-based x-axis, gaps between the segments are minimized. To facilitate the interpretation, the R&S ZNB displays only the limit line segments which provide the limit check criterion (the "tighter" limit line at each point). In the example below, this rule results in a single, continuous lower limit line.



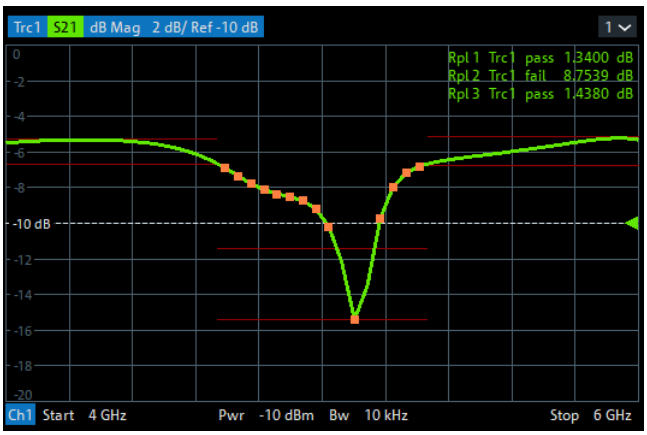
4.4.1.2 Rules for ripple test definition

The analyzer places few restrictions on the definition of ripple limit ranges.

The following rules ensure a maximum of flexibility:

- Ranges do not have to be sorted in ascending or descending order (e.g. the "Start Stimulus" value of range no. n does not have to be smaller than the "Start Stimulus" value of range no. n+1).
- Overlapping ranges are allowed. The limit check in the overlapping area is related to the tighter limit (the pass test involves a logical AND operation).
- Gaps between ranges are allowed and equivalent to switching off an intermediate ripple limit range.
- Ripple limit ranges can be partially or entirely outside the sweep range, however, the limits are only checked at the measurement points.

The following figure shows a ripple limit test involving 3 active ranges.

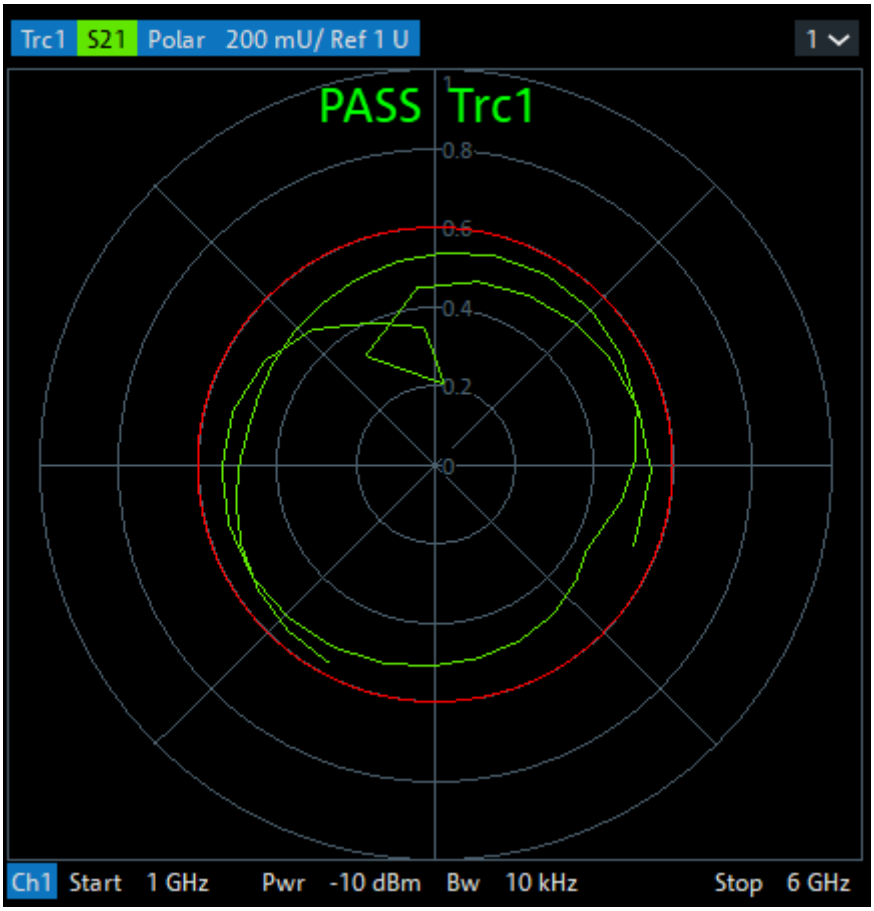


The limit line rules for logarithmic sweeps and segmented frequency sweeps with point-based x-axis also apply to ripple limit lines (see Section 4.4.1.1, "Rules for limit line definition", on page 136).

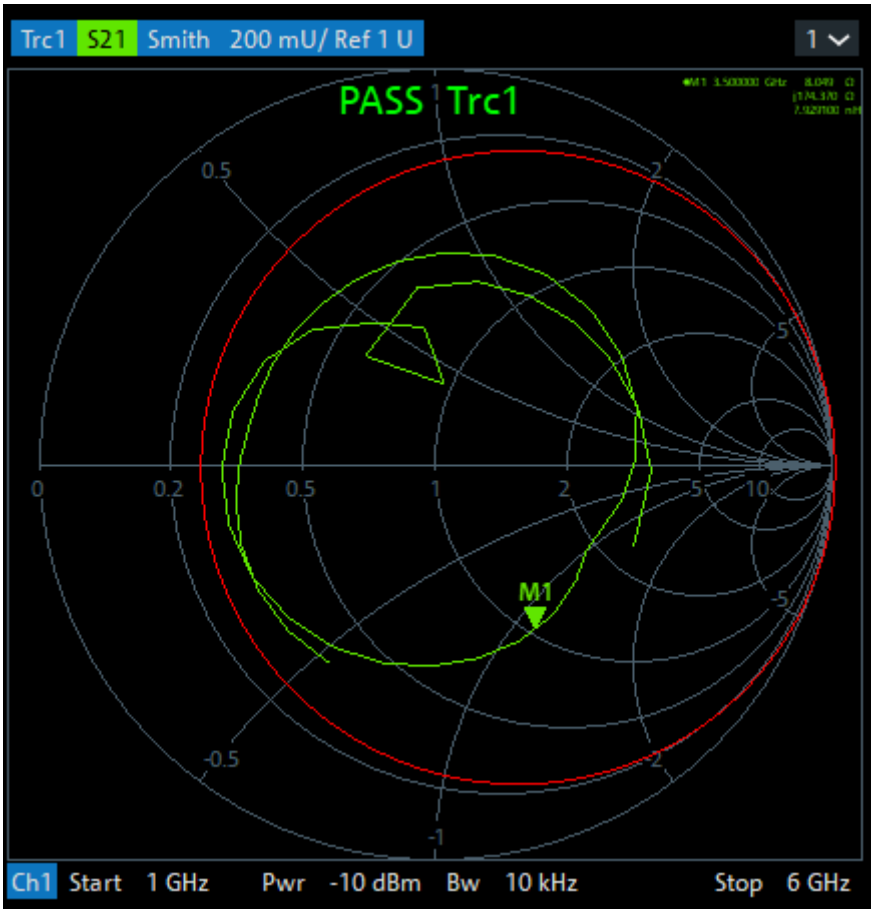
4.4.1.3 Circle limits

A circle limit is a special type of **upper** limit line which is defined by its center coordinate in the diagram and its radius. Depending on the diagram type, circle limit can serve different purposes:

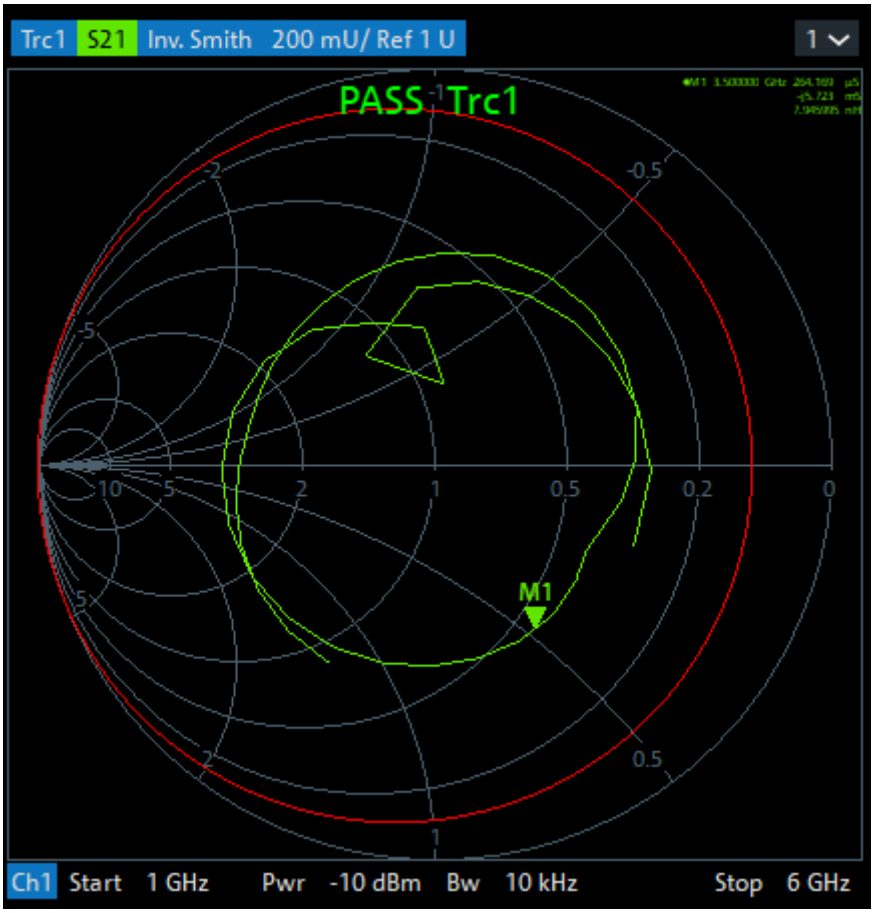
- With a circle limit line centered on the origin of a polar diagram, you can check whether the magnitude of the measurement results exceeds a limit, defined by the radius of the limit line.



- With a circle limit line adjusted to the right border of a Smith diagram (Z = infinity), you can check whether the imaginary part of the impedance (Im(Z), reactance) falls below a limit.



- With a circle limit line centered on the left border of an inverted Smith diagram (Y = infinity), you can check whether the imaginary part of the admittance (Im(Y), susceptance) falls below a limit.



4.4.1.4 File format for limit lines

The analyzer uses a simple ASCII format to export limit line data. By default, the limit line file has the extension `*.limit` and is stored in the directory shown in the "Save Limit Line" and "Recall Limit Line" dialogs. The file starts with a preamble containing the channel and trace name and the header of the segment list. The following lines contain the entries of all editable columns of the list.

Example of a limit line file

The limit line:

	Type	Start Stimulus	Stop Stimulus	Response	Interpolation
1	Upper	9 kHz	8.5 GHz	-40 dB ... -40 dB	Lin
2	Lower	9 kHz	6 GHz	-40 dB ... -40 dB	Lin
3	Off	6 GHz	8.5 GHz	-40 dB ... -40 dB	Lin

is described by the limit line file:

```
# Version 1.00
#
# Channel 1
# Trace 1
en:Type,Start Stimulus[MHz],Stop Stimulus[MHz],Start Response[dB],Stop Response[dB],
Upper,8.9999999999999E-003,8.500000000000000E+003,0.000000000000000E+000,0.000000000000000E+000,
Lower,8.999999999999999E-003,6.000000000000000E+003,-2.000000000000000E+001,-1.000000000000000E+001,
Off,6.000000000000000E+003,8.500000000000000E+003,-1.000000000000000E+001,-1.000000000000000E+001,
```

Compatibility with other instruments

The VNAs of the R&S ZNx and R&S ZVx families use the same file format. Limit line files can be interchanged without restriction.

4.4.1.5 File format for ripple limits

The analyzer uses a simple ASCII format to export ripple limits. By default, the ripple limit file has the extension `*.ripple` and is stored in the directory shown in the "Save Ripple Limits" and "Recall Ripple Limits" dialogs. The file starts with a preamble containing the channel and trace name and the header of the range list. The following lines contain the entries of all editable columns of the list.

Example of a ripple limit file

The ripple limit list:

	Range On/Off	Start Stimulus	Stop Stimulus	Ripple Limit
1	On	3 GHz	5 GHz	11.4535 dB
2	On	6 GHz	6.5 GHz	0 dB
3	On	6.5 GHz	8.5 GHz	0 dB

is described by the ripple limit file:

```
# Version 1.00
#
# Channel 1
# Trace 1
bo:Range On/Off,Start Stimulus[MHz],Stop Stimulus[MHz],Ripple Limit[dB],
true,3.000000000000000E+003,5.000000000000000E+003,1.200000000000000E+001,
true,4.500000000000000E+003,5.500000000000000E+003,5.000000000000000E+000,
false,5.500000000000000E+003,6.000000000000000E+003,4.382581058243317E+000,
```

Compatibility with other instruments

The VNAs of the R&S ZNx and R&S ZVx families use the same file format. Ripple limit files can be interchanged without restriction.

4.4.2 Trace files

The R&S ZNB can store one or several data or memory traces to a file or load a memory trace from a file.

Trace files are ASCII files with selectable file format. The analyzer provides several types of trace files:

- Touchstone (`*.s<n>p`) files
- ASCII ("*.csv") files
- Matlab ("*.dat") files are ASCII files which can be imported and processed in Matlab.

The trace file formats complement each other; see Section 4.4.2.3, "Finding the best file format", on page 152.

- When exporting traces to a file, it is recommended to set the analyzer to single sweep mode (Channel – [Sweep] > "Sweep Control" > "All Channels on Hold"). This mode ensures that a complete sweep is exported.
- The number of decimal places of stimulus and data values in export files is configurable.

4.4.2.1 Touchstone files

The Touchstone file format is a standard for the transfer of frequency-dependent network data. Based on information from Agilent Corporation (the originator of Touchstone), formal specifications were produced by the EIA/IBIS Open Forum.

- Touchstone File Format Specification Rev 1.1
- Touchstone File Format Specification Version 2.0

The R&S ZNB supports Touchstone file versions 1.1 and 2.0.

By convention, Touchstone files use file extensions `.snp`, where `n` is the number of network ports. The following displays an `s1p` export file in "Version 1.1 (ZNx)" format:

```
1  #  HZ   S   RI   R     50.00
2  ! Rohde & Schwarz Vector Network Analyzer
2  ! Rohde-Schwarz,ZNB8-4Port,1311601044100019,3.20
3  ! Created: UTC 12/24/2020, 5:20:26 PM
4  ! My Comment
5  ! freq[Hz]                re:S11                  im:S11
6   1.000000000000000E5    -4.897128641605377E-1    3.767784312367439E-2
    4.259950000000000E7    -5.450598597526550E-1    3.608805686235428E-2
    ...
```

1 = Option line
2 = VNA identification comment
3 = Timestamp comment
4 = User-defined comment
5 = Network data table header comment
6 = Network data table

General syntax rules

Among others, the following general syntax rules apply:

- Touchstone files are ASCII files, 8-bit encoded using ISO/IEC 8859-1:1998. Only the graphical characters 0x20 to 0x7E, the tab character 0x09, and the three line termination sequences LF (0x0A), CR+LF (0x0D + 0x0A), and CR (0x0Dh) are allowed.
- Touchstone files are case-insensitive.
- Blank lines are permitted.

Option line

Each Touchstone file must contain an option line with the following structure:

`# <frequency unit> <parameter> <data format> R <reference resistance>`

- `#` indicates the beginning of the option line
- `<frequency unit>` can be either `Hz`, `kHz`, `MHz` or `GHz`. Default is `GHz`.
  The R&S ZNB always uses `HZ` for exported data.
- `<parameter>` specifies what kind of network parameter data is contained in the file. Default is `S` for S-parameters, which is the only parameter type the R&S ZNB supports.
- `<data format>` specifies how the complex network parameter data are formatted. Possible values are `RI` for "Real-Imag", `MA` for "Lin Mag-Phase", and `DB` for "dB Mag-Phase". Default is `MA`.
  According to the Touchstone file standards, dB Mag = 20 × $\log_{10}$Mag, and the phase is always represented in degrees.
  During data export, the analyzer allows you to select the suitable `<data format>`.
- `<reference resistance>` specifies the impedance system underlying the network data, given as a real, positive resistance. Its value is given in Ω (default 50).
  During export, if the impedances of all involved ports are identical, the analyzer sets the `<reference resistance>` to the real part of this impedance. Otherwise it sets it to 50.

With the exception of the opening `#` symbol and the value following `R`, option line parameters can appear in any order. If a parameter is missing, the default value is assumed.

Comments

Comments can be used for any kind of documentation purpose, e.g. to describe the network parameter data and how they were generated.

Syntactically, comments are preceded by an exclamation mark and terminated by a line termination sequence (see "General syntax rules" on page 144). They can appear anywhere in the file, either on a separate line, or after the parsed content of a line. Any number of comment lines can be inserted.

The content of the comments is not standardized in any way. The following sections describe the comments the R&S ZNB inserts into a Touchstone file during export.



During import, the R&S ZNB ignores all comments.

VNA identification

These comment lines identify the VNA that was used to generate the file.

```
! Rohde & Schwarz Vector Network Analyzer
! Rohde-Schwarz,ZNB8-4Port,1311601044100019,3.20
```

The first line is fixed, the second reflects the configurable identification string of the instrument.

Timestamp

The timestamp comment line reflects the time at which the Touchstone file was created. For "Version 1.1 (ZNx)" export format, it is represented as Coordinated Universal Time (UTC):

```
! Created: UTC 12/24/2020, 5:20:26 PM
```

The other export formats "Version 1.1" and "Version 2.0" use local time (Greenwich Mean Time + offset):

```
! Created: GMT+1 2020-12-24 18:20:26
```

Renormalization information

If, during export, the S-parameter data were renormalized with port-specific impedances, these impedances are written to the output file as a comment. In case the port-specific impedances are different, a note points out that the impedance system underlying the data is not interpreted correctly during import:

```
! The following Port Impedance Renormalization has been used when saving the data.
! PortZ  Port1:100+j0    Port2:50+j0
!
! Note: The Port Impedances differ from the reference impedance of this file.
!       While reading the file the reference impedance value of the option line above
!       is always used.
```

For more information, see "Renormalization of S-parameters" on page 150.



For the "Version 2.0" export format, the port impedances are currently not written to the renormalization information comment. Instead, the real parts of the port impedances appear as arguments of the `[Reference]` keyword.

Network data table header

These comment lines indicate the content of the Network parameter data table right below it.

As the first column of the data table is always designated to the frequency points, the first table header comment always starts with `! freq[Hz]`.

- **"Version 1.1 (ZNx)" export format**
  For this format, the header explicitly specifies the exported parameters and their data format. The port numbers of single-ended S-parameters in the header are the "real" (physical) port numbers:
  `! freq[Hz] re:S22 im:S22 re:S12 im:S12 re:S21 im:S21 re:S11 im:S11`
  Up to port reordering, which is possible via "snp free config" export and via logical port assignment, the order in which the parameter data occur is the one described in the Touchstone 1.1 standard. See "Network parameter data" on page 148.
  If balanced (and mixed mode) parameters are exported, the assigned logical port numbers are used:
  `! freq[Hz] re:Sdd11 im:Sdd22 re:Scd22 im:Scd22 re:Sdc22 im:Sdc22 re:Scc22 im:Scc22`

- **"Version 1.1" and "Version 2.0" export formats**
  These export formats always use single-ended identifiers `Sij` and port numbers 1 to n in the table header.
  `! freq[Hz] S11[Re] S11[Im] ...`
  The mapping between nominal (single-ended) and actual – possibly balanced – ports is given in the port assignments comments.

Port assignments (Version 1.1 and Version 2.0)

For export format "Version 1.1 (ZNx)", the assignments are given in the Network data table header comment lines. For export formats "Version 1.1" and "Version 2.0", they are declared in a dedicated comments section:

```
! Port Assignments:
! Nominal Port -> Actual Test Port
! --------------------------------
!   1 -> 2
!   2 -> 1
```

For balanced ports, the port assignment looks like in the following example:

```
! Port Assignments:
! Nominal Port -> Actual Test Port
! --------------------------------
!   1 -> D1,2
!   2 -> C1,2
```

The "s1p Active Trace" export, which allows to dump an arbitrary S-parameter trace to an `s1p` file, explicitly mentions the exported parameter:

```
! Trace info:
! Nominal Param -> Actual Exported Buffer
! ---------------------------------------
! S11 -> S21
```

User comment

The analyzer's trace export function allows you to insert a custom comment line into the export file.

```
! My Comment
```

Keywords (version 2.0 only)

Keywords are enclosed in square brackets, "[" and "]", and start at the beginning of a line. Arguments after keywords (if any) are separated from the closing bracket and from each other by at least one blank (or tab).

```
[Keyword] argument1 argument2 ...
```



Keywords are new in Touchstone standard version 2.0. They are not allowed in version 1.1 files.

The purpose of the keywords is to give more details about the file contents in a standardized way. During "Version 2.0" export, the R&S ZNB writes the following keywords to the generated Touchstone file:

Keyword	Example	Description
`[Version]`	`[Version] 2.0`	Specification version, typically the first line in the file Currently only `2.0` is allowed
`[Number of Ports]`	`[Number of Ports] 2`	The `n` in the generated `snp` file. Except for the "s1p Active Trace" export, `n` equals the number of physical ports involved in the export.
`[Two-Port Data Order]`	`[Two-Port Data Order] 21_12`	Specifies the data order in `s2p` files. • `21_12` indicates data order `f N11 N21 N12 N22` • `12_21` indicates data order `f N11 N12 N21 N22` (see "Data arrangement" on page 149) The R&S ZNB generates `s2p` files with fixed data order `21_12` (according to the TS1.1 standard).
`[Number of Frequencies]`	`[Number of Frequencies] 201`	Number of exported frequency sweep points; can also be derived from the Network parameter data block.
`[Reference]`	`[Reference] 50 75`	If the exported S-parameter data were renormalized with port-specific impedances, the real parts of these impedances are listed as arguments of the `[Reference]` keyword. Overrides the common reference impedance of the "Option line" on page 144 **Note:** currently ignored during import
`[Mixed-Mode Order]`	`[Mixed-Mode Order] D1,2 C1,2`	Required if a balanced port is involved. In the example to the left, nominal port 1 is the differential mode and nominal port 2 is the common mode of balanced port (1,2). Conveys the same information as the Port assignments (Version 1.1 and Version 2.0) comment
`[Network Data]`	`[Network Data] ! no args`	Marks the begin of the Network parameter data block
`[End]`	`[End] ! no args`	Marks the end of the network parameter data section and is typically placed in the last line of the export file

For details and more keywords, see the Touchstone File Format Specification Version 2.0 (http://www.ibis.org/touchstone_ver2.0/touchstone_ver2_0.pdf).

During import, the R&S ZNB currently only considers the `[Two-Port Data Order]`.

Network parameter data

The network parameter data section contains one block of network parameters per frequency point. Each block is preceded by a frequency value, i.e. the first entry in the first (or only) data line of a data block is a frequency value. The complex network parameter data is formatted as pairs of values.

The following general rules apply:

- Individual entries in a network data line are separated by whitespace
- In version 1.1 files, no more than four parameters are allowed per network data line. No restriction exists on the number of data pairs on a line in version 2.0 files.
- The network data blocks are arranged in ascending order of frequency



For segmented sweeps, the R&S ZNB makes an exception to the strict "ascending order" principle. During export, the segments are always kept together, which can result in downward "frequency jumps" at segment borders.

Data arrangement

During export, for each block, the R&S ZNB arranges the frequency value f and network parameter data $(Nij)_{1\le i,j\le n}$ as described in the Touchstone 1.1 standard.

- **n = 1**
  ```
  f N11
  ```
- **n = 2**
  ```
  f N11 N21 N12 N22
  ```
- **n = 3, 4**
  ```
  f N11 N12 N13
    N21 N22 N13
    N31 N32 N33
  ```
  Similar matrix arrangement for n=4.
- **n > 4**
  For n > 4, the "max. four parameters per data line" restriction applies. It is overcome by wrapping the matrix rows at multiples of four network parameter values.
  For n = 9:
  ```
  f N11 N12 N13 N14
    N15 N16 N17 N18
    N19
    N21 ...
    N95 N96 N97 N98
    N99
  f N11 N12 N13 N14
    N15 N16 N17 N18
    N19
    N21 ...
    N95 N96 N97 N98
    N99
  ```

The data blocks are arranged in ascending order of frequency.



The "snp Free Config" export mode allows you to define:

- The set of (physical) ports whose S-parameters shall be exported
- The order in which they are exported

The examples above represent the simple scenario with consecutive ports {1,...,n} and natural ordering.

Data format

Frequencies are represented as single positive numbers, network parameter data as pairs of numbers. The <frequency unit> and <data format> are specified in the Option line. Numbers can be represented as decimal integers or floating point numbers, with "." used as decimal separator. Scientific notation is allowed (e.g., 1.2345e-12). No minimum or maximum limits are placed on numerical precision.

For Touchstone export, the R&S ZNB uses the normalized scientific notation, with a significand (mantissa) of length 17 (+1 for the minus sign) and an exponent with maximum length 2 (+1 for the minus sign).

```
1.000000000000000e+09    -2.375070438098596e-01    -4.533104459856211e-01
1.010000000000000e+09    -3.132740349817996e-01    -5.874596585157938e-01
1.020000000000000e+09    -3.062444919836442e-01    -4.587379501906624e-01
```

Conditions for Touchstone file export

- One-port Touchstone files with data from a single trace
  Typically, Touchstone files contain a complete set of S-parameter traces of an n-port network. The only exceptions are s1p files created from arbitrary S-parameter traces using the "s1p Active Trace" export function in the Trace Data tab or the remote command MMEMory:STORe:TRACe.
- Touchstone files containing S-matrices
  - For a one-port Touchstone file, the reflection coefficient for the specified port (S_{ii} for port i) must be measured. If a full one-port (Refl OSM) or a full n-port (TOSM, ...) calibration is available for the specified port, it is possible to export the data even when the trace is not displayed.
  - For a multiport Touchstone file *.s<n>p, either a full multiport system error correction or a complete set of n^2 S-parameter traces must be available.
    If the port configuration contains balanced ports, the exported Touchstone file contains the converted single-ended S-parameters unless you enable "Balanced Params" during export.
  - Export of balanced (symmetric) S-parameters must be enabled. If you want to export balanced S-parameters, make sure that all these parameters are measured, and that for each involved balanced port both physical ports are part of the export.

  This type of export can be initiated from the GUI by using the "s<n>p Port ..." or "snp Free Config." functions of the Trace Data tab or by the remote command MMEMory:STORe:TRACe:PORTs on page 1299.

Renormalization of S-parameters

Renormalization means that the S-parameters at connector impedances are converted to S-parameters at certain target impedances.

During Touchstone export, the S-parameters can be renormalized in two ways:

- Common target impedance
  The <reference resistance> of the Option line is taken as common target impedance and the data is renormalized to the common target impedance, regardless of the reference impedances of the involved ports.

If the impedances of all involved ports are identical, the analyzer sets the <reference resistance> in the Option line to the real part of this impedance. Otherwise it sets it to 50 (Ω).

- Port-specific Reference impedances
  In this case, the reference impedances of the individual ports are used for the renormalization and these impedances are documented in the Renormalization information comment. If the port-specific reference impedances are different, an additional warning is added to this comment, indicating that the option line contains a non-matching reference resistance.
  Note that when reimporting this type of file into standard applications (including the R&S ZNB itself), the <reference resistance> of the Touchstone option line is used and the impedance system underlying the data is not interpreted correctly.

4.4.2.2 ASCII (*.csv) files

An ASCII file contains a header and the actual trace data:

freq;reTrc1_S21;imTrc1_S21;reMem2[Trc1]_S21;imMem2[Trc1]_S21;

300000.000000;0.000000;0.000000;0.000000;0.000000;

40499497.487437;0.000000;0.000000;0.000000;0.000000;

80698994.974874;0.494927;-0.065174;0.500833;-0.074866;

120898492.462312;0.497959;-0.111724;0.488029;-0.107375;

...

The header consists of the following data elements:

- <Stimulus> stimulus variable: freq for frequency sweep, power for power sweep, time for time sweep, trigger for CW mode sweep.
- <reTrace1> first response value of first trace: re<Trace_Name>, mag<Trace_Name> or db<Trace_Name> for output format Re/Im, lin. Mag-Phase or dB Mag-Phase, respectively. The data format for export files can be selected in the Export Data dialog.
- <imTrace1> second response value of first trace: im<Trace_Name> for output format Re/Im, ang<Trace_Name> for output formats lin. Mag-Phase or dB Mag-Phase. The data format for export files can be selected in the Export Data dialog.
- <reTrace2> first response value of second trace: re<Trace_Name>, mag<Trace_Name> or db<Trace_Name> for output format Re/Im, lin. Mag-Phase or dB Mag-Phase, respectively. The data format for export files can be selected in the Export Data dialog.
- <imTrace2>... second response value of second trace: im<Trace_Name> for output format Re/Im, ang<Trace_Name> for output formats lin. Mag-Phase or dB Mag-Phase. The data format for export files can be selected in the Export Data dialog. first response value of second trace. HZ / KHZ / MHZ / GHZ allowed for imported files. The analyzer always uses HZ for exported data. second response value of first trace: im<Trace_Name> for output format Re/Im, ang<Trace_Name> for output formats lin. Mag-Phase or dB Mag-Phase. The data format for export files can be selected in the Export Data dialog.

The trace data is arranged as described in the header. Different values are separated by semicolons, commas or other characters, depending on the selected "Decimal Separator" in the "Export ... Data" dialogs. A semicolon is inserted before the end of each line.

The stimulus values are arranged in ascending order.

4.4.2.3 Finding the best file format

The file format depends on how you want to use the exported data.

Use a Touchstone file format to export S-parameter data traces to a file that can be evaluated with applications such as Keysight's Advanced Design System (the former Microwave Design System), and to convert mixed mode parameters back to single-ended parameters. The data must be acquired in a frequency sweep. Note the "Conditions for Touchstone file export" on page 150.



Touchstone files cannot be used to export mathematical traces.

Use the ASCII (*.csv) format if you want to do one of the following:

- Import the created file into a spreadsheet application such as Microsoft Excel.
- Export an arbitrary number of traces, multiple traces with the same parameter or memory traces.
- Export traces acquired in a power sweep or CW sweep.
- Export time domain traces
- Use export options.

Use the **Matlab (`*.dat`)** format if you want to import and process the trace data in Matlab.

4.4.3 Memory-mapped trace data transfer

The VNA firmware uses the *Named Shared Memory* Windows feature to share trace data with other processes running on the instrument. Such processes or applications can read trace data directly from the shared memory, which is much faster than using SCPI commands.

Multiple traces can be written to the same shared memory buffer. For each of these traces, the firmware stores a configurable number of formatted or unformatted trace values to a contiguous part of the shared memory.

- A formatted (real) trace value takes up 8 bytes of memory.
- An unformatted (complex) trace value takes up 16 bytes: 8 bytes for the real part, 8 bytes for the imaginary part.

Memory-mapped trace data transfer is a pure remote control feature. Once set up, the VNA firmware refreshes the shared memory trace data with every sweep. Other pro-

cesses can read these data via standard APIs for Windows *Named Shared Memory* that are available for various programming languages.



You can create multiple shared memory buffers, e.g. with different intended life span.

Set up a shared memory buffer

The setup of a shared memory buffer and the allocation of trace data is performed by a sequence of SYSTem:DATA:MEMory... commands, starting with an `INITialize` and finished by a `COMMit`.

1. Initialize the shared memory buffer:
   `SYSTem:DATA:MEMory:INITalize`
2. Add traces:
   a) `SYSTem:DATA:MEMory:ADD '<TrName1>', FDATA|SDATa, #Values1`
   Select an existing trace name, choose whether to use formatted trace data (`FDATa`) or unformatted trace data (`SDATa`), and define how many trace values the firmware writes to the buffer per sweep.
   The data block of the first trace has an offset of 0 bytes relative to the start of the shared memory buffer.
   b) `SYSTem:DATA:MEMory:ADD '<TrName2>', FDATA|SDATa, #Values2`
   Use `SYSTem:DATA:MEMory:OFFSet?` to query the offset in bytes of the second trace data block. Processes that want to read the trace data from the buffer need to know this offset. Its size depends on whether the first trace data block contains formatted (real) or unformatted (complex) values.
   c) ...
   Use `SYSTem:DATA:MEMory:OFFSet?` to query the offset of the last added trace.
3. Create the named shared memory buffer:
   `SYSTem:DATA:MEMory:COMMit '<ShMemName>'`
   The name must be unique. Before the `COMMit` you can use `SYSTem:DATA:MEMory:CATalog?` to query the names of existing shared memory buffers.
   The size of the last created shared memory buffer can be queried using `SYSTem:DATA:MEMory:SIZE?`.

Use a shared memory buffer

After you have created a named shared memory buffer, you can access it – from the instrument – using your favorite programming language and API. With Python, for instance, you can use the `mmap` module for this purpose.

Delete a shared memory buffer

To delete a shared memory buffer and free the occupied memory, you can use one of the following commands.

1. `SYSTem:DATA:MEMory:DELete '<ShMemName>'`
   deletes a single shared memory buffer by name.
2. `SYSTem:DATA:MEMory:RESet`
   deletes all shared memory buffers created using `SYSTem:DATA:MEMory:COMMit`.

4.5 Calibration

Calibration or system error correction is the process of eliminating systematic, reproducible errors from the measurement results (S-parameters and derived quantities; see Section 4.1.5, "Data flow", on page 89). The process involves the following stages:

1. A set of calibration standards is selected and measured over the required sweep range.
   For many calibration types, the magnitude and phase response of each calibration standard (i.e. its S-parameters if no system errors occur) must be known within the entire sweep range. In some calibration procedures (TRL, TNA, TRM), part of the characteristics of the standards can be auto-determined due to implicit redundancy (self-calibration).
2. The analyzer compares the measurement data of the standards with their known, ideal response. The difference is used to calculate the system errors using a particular error model (calibration type) and derive a set of system error correction data.
3. The system error correction data is used to correct the measurement results of a DUT that is measured instead of the standards.

Calibration is always channel-specific because it depends on the hardware settings, in particular on the sweep range. This means that a system error correction data set is stored with the calibrated channel.

The analyzer provides a wide range of sophisticated calibration methods for all types of measurements. Which calibration method is selected depends on the expected system errors, the accuracy requirements of the measurement, on the test setup and on the types of calibration standards available.

Due to the analyzer's calibration wizard, calibration is a straightforward, menu-guided process. Moreover, it is possible to perform the entire calibration process automatically using a Calibration Unit (e.g. R&S ZN-Z5x or R&S ZN-Z15x).

The system error correction data determined in a calibration procedure are stored on the analyzer. You can read these correction data using the remote control command `[SENSe<Ch>:]CORRection:CDATa`. You can also replace the correction data of the analyzer by your own correction data sets.

Cal Off label

A label "Cal Off" appears in the trace line if the system error correction no longer applies to the trace:



This can happen for one of the following reasons:

- The sweep range is outside the calibrated frequency range.
- The channel calibration is not sufficient for the measured quantity (e.g. a one-port calibration has been performed, but the measured quantity is a transmission parameter).
- There is a mismatch between the current Frequency Offset settings and the Frequency Offset settings that were used during calibration (see "Parallel measurements with frequency offset" on page 86)
- There is a mismatch between the current sweep mode(s) and the sweep mode(s) that were used during calibration (see Section 4.1.4.5, "Stepped vs. swept mode", on page 88)
- The system error correction has been switched off deliberately ("User Cal Active" is disabled).

The analyzer provides other labels to indicate the status of the current calibration; see Section 4.5.4, "Calibration state labels", on page 175.

Calibration and port de-/activation

The analyzer firmware automatically activates/deactivates ports during/after a (successful) calibration:

- Calibrated ports that were previously disabled, are automatically enabled as single-ended logical ports.
- An uncalibrated port that is not used by a measurement (i.e. the port is not required by any trace of the related channel) is disabled.

4.5.1 Calibration types

The analyzer provides a wide range of calibration types for one, two or more ports. The calibration types differ in the number and types of standards used, the error terms, i.e. the type of systematic errors corrected and the general accuracy. The following table gives an overview for the calibration types related to calibration standards.

For calibration types including power calibration, see Table 4-15.

Table 4-7: Overview of calibration types

Calibration Type	Standards	Parameters	Error Terms	General Accuracy	Application
Reflection Normalization	Open or Short	S_{ii} for Port i	Reflection tracking	Low to medium	Reflection measurements on any port.
Transmission Normalization	Through	S_{ij} for port pair (i,j), i ≠ j	Transmission tracking	Medium	Transmission measurements in any direction and between any combination of ports.
Reflection OSM	Open, Short, Match[1)]	S_{ii} for Port i	Reflection tracking, Source match Directivity,	High	Reflection measurements on any port.
One Path Two Ports	Open, Short, Match[1)] (at source port), Through[2)] between the source port and all target ports	$S_{t,j}$ for fixed source port j and target ports t	Reflection tracking, Source match, Directivity, Transmission tracking	Medium to high	Unidirectional transmission measurements in any direction and between any combination of ports.
TOSM or UOSM (n-port)	Open, Short, Match[1)] (at each port), Through[2)] (between all port pairs)	All	Reflection tracking, Source match, Directivity, Load match, Transmission tracking,	High	Reflection and transmission measurements; classical 12-term error correction model.
Adapter Removal (2-port)	Open, Short, Match[1)] (at each port), Through	All Reflection parameters with and without adapter	Reflection tracking, Source match, Directivity, Load match, Transmission tracking,	High	Reflection and transmission measurements; classical 12-term error correction model.
TOM (n-port)	Open, Match (at both ports), Through (between all port pairs)	All	Reflection tracking, Source match, Directivity, Load match, Transmission tracking	High	Reflection and transmission measurements.
TSM (n-port)	Short, Match (at both ports), Through (between all port pairs)	All	Reflection tracking, Source match, Directivity, Load match, Transmission tracking	High	Reflection and transmission measurements.

Calibration Type	Standards	Parameters	Error Terms	General Accuracy	Application
TRM (n-port)	Reflect (equal at both ports), Match, Through (between all port pairs)	All	Reflection tracking, Source match, Directivity, Load match, Transmission tracking	High	Reflection and transmission measurements, especially in test fixtures.
TRL (n-port)	Reflect (at both ports), Through, Line1, other Lines (optional), combination with TRM (optional)	All	Reflection tracking, Source match, Directivity, Load match, Transmission tracking	High, high directivity	Reflection and transmission measurements, especially for planar circuits. Limited bandwidth.
TNA (n-port)	Through, Attenuation, Symmetric network	All	Reflection tracking, Source match, Directivity, Load match, Transmission tracking	High, lowest requirements on standards	Reflection and transmission measurements, especially for planar circuits.

[1] Or any other known one-port standard. To be used in a guided calibration, the known standards must be declared to be Open, Short, and Match irrespective of their properties.

[2] Or any other known two-port standard. To be used in a guided calibration, the known standard must be declared to be Through, irrespective of its properties.

[3] A reciprocal reference mixer, whose LO signal must be generated by the VNA. Rohde & Schwarz offers the calibration mixer R&S ZN-ZM292 for this purpose.

[4] A reference mixer with known group delay characteristics (with or without embedded LO).

The calibration type must be selected in accordance with the test setup. Select the calibration type for which you can obtain or design the most accurate standards and for which you can measure the required parameters with best accuracy.

- Normalization (reflection, transmission) 158
- Reflection OSM calibration 158
- One Path Two Ports calibration 158
- TOSM and UOSM calibration 159
- Adapter removal 160
- TOM calibration 162
- TSM calibration 162
- TRM calibration 163
- TRL calibration 163
- TNA calibration 166
- Full n-Port calibration with reduced number of Through connections 166
- Complementary isolation measurement 168

4.5.1.1 Normalization (reflection, transmission)

A normalization is the simplest calibration type since it requires the measurement of only one standard for each calibrated S-parameter:

- One-port (reflection) S-parameters (S_{11}, S_{22}, ...) are calibrated with an Open or a Short standard providing the reflection tracking error term.
- Two-port (transmission) S-parameters (S_{12}, S_{21}, ...) are calibrated with a Through standard providing the transmission tracking error term.

Normalization means that the measured S-parameter at each sweep point is divided by the corresponding S-parameter of the standard. A normalization eliminates the frequency-dependent attenuation and phase shift in the measurement path (reflection or transmission tracking error). It does not compensate for directivity or mismatch errors. This limits the accuracy of a normalization.



- Manual reflection normalizations offer Complementary Match standard measurements
- Manual transmission normalizations support Complementary isolation measurement (optional).

Complementary Match standard measurements

For reflection normalizations, the mandatory Open or Short measurements can be complemented by optional Match measurements. Additionally measuring a Match standard allows you to eliminate errors due to the directivity of the internal couplers, which improves the accuracy of reflection measurements on well-matched DUTs (high return loss).



For reflection measurements on DUTs with low return loss, accuracy may be degraded compared to a simple reflection normalization.

4.5.1.2 Reflection OSM calibration

A reflection OSM (full one-port) calibration requires a Short, an Open and a Match standard to be connected to a single test port. The three standard measurements are used to derive all three reflection error terms:

- The Short and Open standards are used to derive the source match and the reflection tracking error terms.
- The Match standard is used to derive the directivity error.

A reflection OSM calibration is more accurate than a normalization but is only applicable for reflection measurements.

4.5.1.3 One Path Two Ports calibration

A one path two ports calibration combines a reflection OSM (full one-port) calibration with a transmission normalization. The fully calibrated port is termed the node port.

This calibration type requires a Short, an Open and a Match standard to be connected to a single test port plus a Through standard between this calibrated source port and the other load ports. The four standard measurements are used to derive the following error terms:

- The Short and Open standards are used to derive the source match and the reflection tracking error terms at the source port.
- The Match standard is used to derive the directivity error at the source port.
- The Through standard provides the transmission tracking error terms.

For calibration of two ports, a One Path Two Ports calibration requires only four standards to be connected (instead of 7 for a full two-port TOSM calibration). It is suitable when only the forward (e.g. S_{11} and S_{21}) or reverse S-parameters (e.g. S_{22} and S_{12}) are needed, and if the DUT is well matched, especially at the load port.

A One Path Two Ports calibration is also the best calibration method for test setups with unidirectional signal flow.

4.5.1.4 TOSM and UOSM calibration

TOSM

A TOSM (Through – Open – Short – Match) calibration requires the same standards as the one path two ports calibration, however, all measurements are performed in the forward and reverse direction. TOSM is also referred to as SOLT (Short – Open – Load = Match – Through) calibration. The four standards are used to derive 6 error terms for each signal direction:

- In addition to the source match and reflection tracking error terms provided by the one-path two-port calibration, TOSM also provides the load match.
- The directivity error is determined at all source ports.
- The transmission tracking is determined for each direction.

TOSM calibration is provided for arbitrary n-port measurements (n>1). The number of required standard measurements and of error terms increases as shown in the following table.

Number of ports	Number of standards to be connected	Number of standard measurements	Number of error terms*)
2	2 * 3 +1 = 7	2 * 3 +2 * 1 = 8	2 * 3 + 2 * 2 = 10
3	3 * 3 + 2 + 1 = 12	3 * 3 +2 * (2 + 1) = 15	3 * 3 + 2 * 2 * 3 = 21
4	4 * 3 +3 + 2 + 1 = 18	4 * 3 +2 * (3 + 2 + 1) = 24	4 * 3 + 2 * 2 * 6 = 36

*) No isolation terms are available.

An Open, Through and Match measurement is required at each port; in addition, for a full TOSM, a Through must be measured for every directed port pair. Therefore the

total number of standard measurements for an n-port TOSM calibration is *3n + n (n-1) = n (n+2)*. However, this number can be significantly decreased without losing too much precision (see Section 4.5.1.11, "Full n-Port calibration with reduced number of Through connections", on page 166).

The analyzer automatically performs each through measurement in both directions, so the number of connected standards is smaller than the number of measurements.

Manual TOSM calibration supports Complementary isolation measurement (optional).

UOSM: TOSM with unknown Through

The analyzer can perform a TOSM calibration with any 2-port network serving as through connection, as long as it fulfills the reciprocity condition $S_{21} = S_{12}$. The modified TOSM calibration is referred to as UOSM (Unknown through – Open – Short – Match) calibration. It can be selected as follows:

- If different connector types are assigned to the test ports, the analyzer automatically replaces TOSM by UOSM.
  [The network analyzer supports different connector types at its test ports to measure DUTs with different port connectors; see also Section 4.5.1.5, "Adapter removal", on page 160.]
- If the same connector types are used but an appropriate Through standard is not defined, the analyzer also replaces TOSM by UOSM.
- UOSM can be selected explicitly in the "Calibration Setting" dialog.

After acquiring the calibration sweep data for the unknown through, the analyzer automatically determines its delay time/transmission phase.

4.5.1.5 Adapter removal

Many DUTs use different connector types on their RF ports (e.g. port 1: N-type connector, female; port 2: PC 3.5-type connector, female).

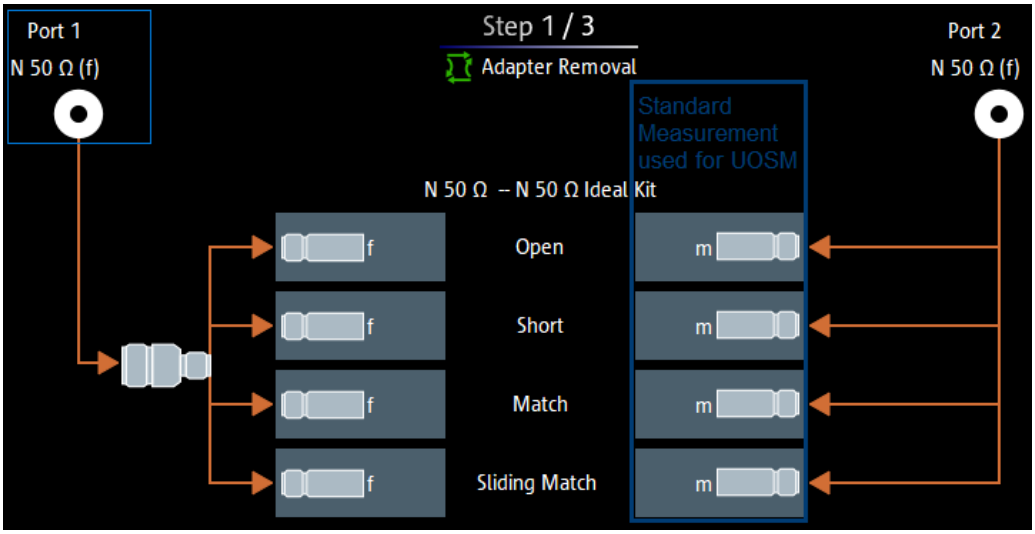
DUTs of this sort are also known as **noninsertable devices**.

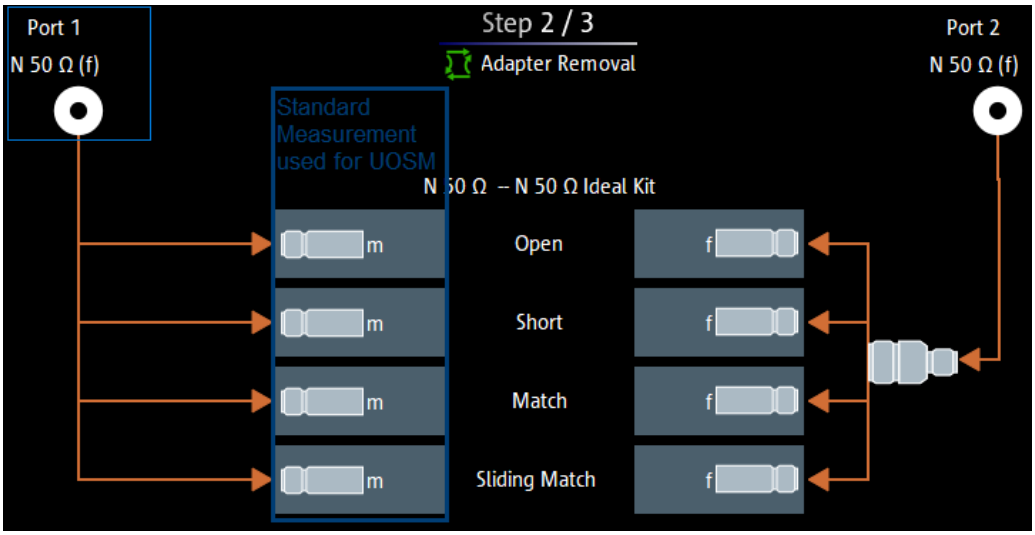
A suitable calibration kit is then required for each of the different connector types. The kit must include, as a minimum, the standards that are used for one-port calibration. The through-connection between test ports with different connector types must be made using **adapters**.

The problem here is that unlike Through standards, adapters are usually not characterized, i.e. their delay time/transmission phase is unknown.

If the adapter fulfills the reciprocity condition $S_{21} = S_{12}$, it can serve as **U**nknown through in an **U**OSM calibration (see "UOSM: TOSM with unknown Through" on page 160).

Adapter Removal is an extension of the **2-port** UOSM calibration. It requires two additional reflection OSM calibrations with the adapter successively connected to port 1 and port 2.





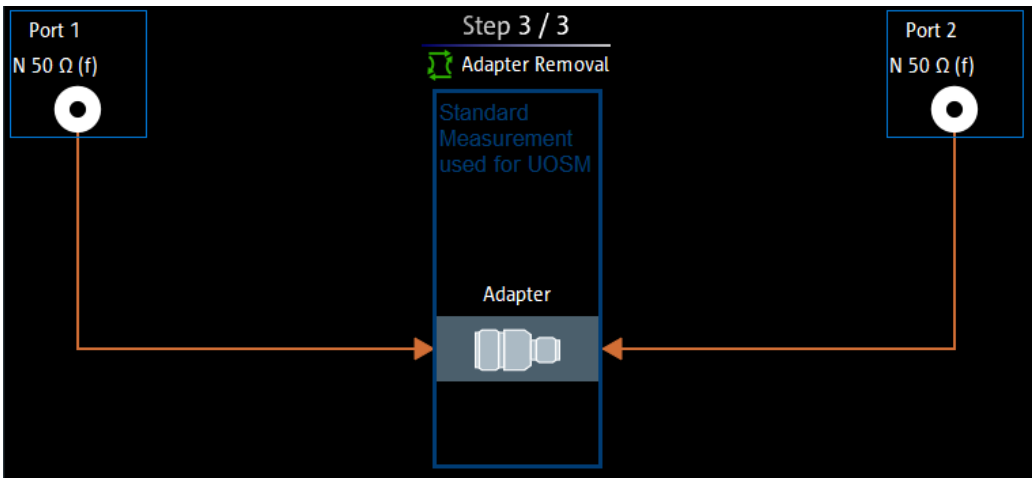


Figure 4-9: Adapter Removal vs. UOSM

The obtained adapter characteristics are mathematically removed from the obtained error coefficients. Uncertainties arising from a non-ideal characterization of the unknown through almost cancel, whereas they add up in the UOSM technique. As a consequence, Adapter Removal will provide more accurate results.

This holds up to a transition point where uncertainties due to connectivity issues increase.

- Adapter Removal is not defined for more than 2 ports. However, with "Multiple Calibrations per Channel" enabled, multiple (disjoint) port pairs can be calibrated using Adapter Removal.
- Currently Adapter Removal is not supported with Automatic calibration.

4.5.1.6 TOM calibration

A TOM (Through – Open – Match) calibration requires a low-reflection, low-loss Through standard with an electrical length that can be different from zero, an Open, and a Match. The characteristics of all standards must be fully known; the Match can have non-ideal characteristics.

4.5.1.7 TSM calibration

A TSM (Through – Short – Match) calibration requires a low-reflection, low-loss Through standard with an electrical length that can be different from zero, a Short, and a Match. The characteristics of all standards must be fully known; the Match can have non-ideal characteristics.

TSM calibration can replace TOM calibration if no appropriate Open standard is available, especially in the high frequency domain.

4.5.1.8 TRM calibration

A TRM (Through – Reflect – Match) calibration requires a low-reflection, low-loss Through standard with an electrical length that can be different from zero, a Reflect, and a Match. The magnitude of the reflection coefficient of the Reflect standard can be unknown but must be nonzero; its phase must be roughly known (90 deg). The magnitude and phase of the reflection coefficient must be the same at both test ports.

TRM calibration is especially useful for DUTs in test fixtures.

4.5.1.9 TRL calibration

A TRL (Through – Reflect – Line) calibration requires the two-port standards Through and Line, which are both assumed to be ideally matched. The Through must be lossless, and its length must be exactly known. The length of the Line standard must be known approximately.

Furthermore, a reflecting one-port standard (Reflect) is needed. The magnitude of the reflection coefficient of the Reflect standard can be unknown but must be nonzero; its phase must be roughly known (90 deg). The magnitude and phase of the reflection coefficient must be the same at both test ports.

TRL calibration is especially useful for DUTs in planar line technology (e.g. test fixtures, on-wafer measurements) where it is difficult to design and connect accurately modeled Open, Short or Match standards.

If TRL is not practicable, TNA can be an alternative.

TRL with several Lines and with TRM

The system of equations that have to be solved for the error terms exhibits singularities whenever the length difference ΔL between the Through and the Line is an integer multiple of half of the wavelength:

$$\Delta L \neq n \frac{\lambda}{2}$$

As a rule, singularities are avoided with sufficient accuracy if the phase shift resulting from the (electric) length difference between the Through and the Line standard is between 20° and 160°. This corresponds to a ratio of 1:8 for the start and stop frequency of the calibrated sweep range.

To shift the calibrated sweep range to smaller or larger frequencies, you can use a longer or shorter Line. To extend the calibrated range, use one of the following methods:

- Perform a multiline TRL calibration with more two or more Line standards. With an appropriate length of the Lines, the ratio for the start and stop frequency of the calibrated sweep range can increase to approx. 1:64 (for 2 lines) or 1:512 (for 3 lines).
- Perform a NIST multiline calibration according to:
  - R.B. Marks, "A multiline method of network analyzer calibration", IEEE Transactions on Microwave Theory and Techniques (Volume: 39, Issue: 7, July 1991)
  - D.C. DeGroot, J.A. Jargon, R.B. Marks, “Multiline TRL revealed”, 60th ARFTG Conference Digest, Fall 2002.

  Recommended in particular for on-wafer measurements, and planar line and circuit measurements.
- In the low-frequency domain where TRL becomes inaccurate, replace TRL by TRM calibration. See "Low-frequency extension with TRM" on page 166.

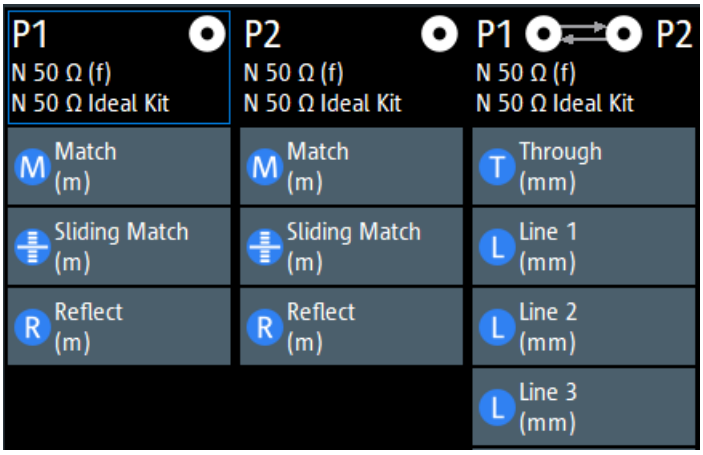
The methods can be combined or used separately. The list of measured standards in the calibration step for TRL calibration is extended if the calibration kit in use contains the necessary standards:

- A multiline calibration requires two or more different Lines of matching gender. For highest precision, measure the Line connections between all pairs of ports to be calibrated.
- A TRM extension at low frequencies requires either a Match or a Sliding Match standard, measured at each port.

The complete list of measured standards for a two-port calibration is shown below.

- For a **TRL calibration with 1 Line**, you have to measure the Reflect standard at both ports, the Through, and one Line standard.
- For a **TRL calibration with n Lines**, measure n-1 Line standards in addition.
- For the TRM extension, you have to measure the Reflect and (sliding) Match standards at both ports (and the Through). See also Section 4.5.2.3, "Sliding Match standards", on page 172.

The analyzer firmware considers a TRL calibration as valid, if the standards for a TRL calibration with 1 Line have been measured. The TRM extensions are applied automatically if the necessary standards have been measured.



Example: TRL calibration with two and three Lines

If several Lines with different lengths are measured, the analyzer automatically divides the calibrated range into segments. The calibration data of the longest line is applied to the lowest segment, the calibration data of the shortest line to the highest segment.

The calibration sweep segments for two Lines with electric lengths l_{long} and l_{short} ($l_{long} > l_{short}$) are obtained as follows (the Through standard is assumed to be of length l_{thr}):

- The longer Line can be used up to a frequency f_{long} where its transmission phase is equal to 160 deg. This frequency is equal to:
  $f_{long} = 4*c_0/[9*(l_{long} - l_{thr})]$.
- The shorter Line can be used from a frequency f_{short} where its transmission phase is equal to 20 deg. This frequency is equal to:
  $f_{short} = c_0/[18*(l_{short} - l_{thr})]$.
- The border between the two frequency segments f_{div} is calculated as the geometric mean of f_{long} and f_{short}, i.e.:
  $f_{div} = sqrt(f_{long} * f_{short})$.

The formulas are also applied if $f_{long} < f_{short}$.

For a TRL calibration using three Lines with different length, the allowed frequency ranges are calculated in an analogous manner to obtain three (ideally overlapping) frequency ranges. The borders between two adjacent frequency ranges are calculated as the geometric mean of the frequency limits f_{long} and f_{short} of the two ranges.

- You do not have to measure all additional Line standards in the list. If the calibrated frequency range is small enough, the calibration is valid when the analyzer has acquired correction data for a single Line standard.
- For an n-port calibration, you do not have to measure all n(n-1)/2 Through and n(n-1)/2 Line connections. The analyzer can calculate the error terms after n-1 Through and n-1 Line measurements, if the measured Throughs/Lines connect all ports to be calibrated.
- The Match and Sliding Match standards are not necessary for TRL calibration. However, they must be measured if TRL is combined with TRM calibration.

Low-frequency extension with TRM

TRL calibration becomes inaccurate if the electrical length difference between Line and Through standard corresponds to a phase shift below 20°. In practice, TRL is only practicable above a threshold frequency $c_0/[18*(l_{long}- l_{thr})]$, where l_{long} denotes the longest electrical length of the used Line standards, and l_{thr} the electrical length of the Through. The analyzer assumes $l_{thr} << l_{long}$ and uses $c_0/(18*l_{long})$ as the frequency threshold.

E.g., for a line with l_{long} = 16.666 cm, the threshold frequency is 100 MHz.

At frequencies below this threshold frequency, TRL calculation is automatically replaced by TRM calculation, if the necessary Match data have been acquired.

Accuracy conditions for the Lines

The length error of the Line, converted into a transmission phase error, must be below the minimum difference to the singularity points 0 deg or 180 deg multiplied by two. Suppose that an approximately known Line standard causes a transmission phase 30 deg at the start frequency and of 160 deg at the stop frequency of the sweep. Its length error must cause a phase difference below (180 deg – 160 deg)*2 = 40 deg.

4.5.1.10 TNA calibration

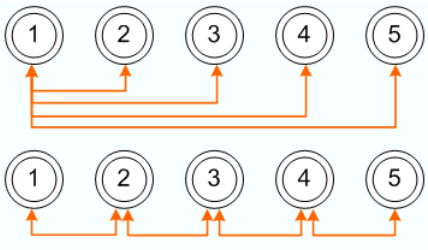
A TNA (Through – Network – Attenuation) calibration requires two-port standards only. Again, the Through standard must be ideally matched and lossless. The Symmetric Network must have the same properties as the Reflect standard used for a TRL calibration. I.e., the magnitude of its reflection coefficient can be unknown but must be nonzero. Its phase must be roughly known (±90 deg). The magnitude and phase of the reflection coefficient must be the same at both test ports. The Attenuation standard must be well matched on both sides and cause an attenuation different from 0 dB; the exact value of the transmission coefficient is not important.

As with TRL, TNA calibration is especially useful for planar DUTs. If TNA is not practicable, TRL can be an alternative.

4.5.1.11 Full n-Port calibration with reduced number of Through connections

The analyzer can calculate the error terms for a full n-port calibration after n−1 Through measurements, if the measured Throughs connect all ports to be calibrated. The correction values for the unmeasured Throughs can then be calculated from the measured ones.

To establish a sufficient set of Through connections, you can select an arbitrary test port as the "center" and measure all Through connections to this test port ("star-shaped calibration"). You can also connect all ports in increasing order, e.g. 1→2, 2→3, 3→4 ...



[For the mathematically inclined: the graph constructed from the calibration ports as nodes and the measured Throughs as edges must be connected.]

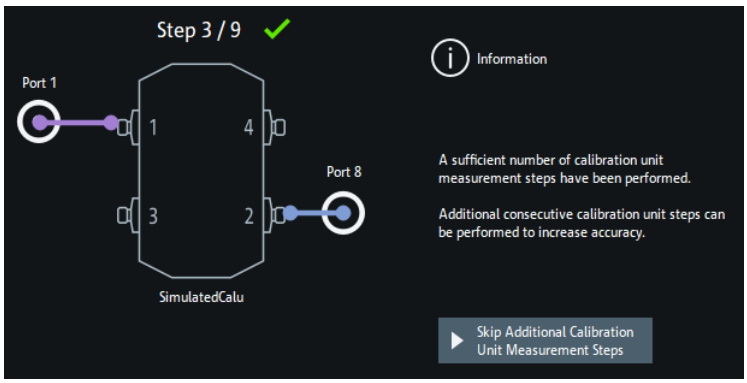
If an external switch matrix is configured and you want to calibrate three or more test ports that are all on the **same submatrix**, then an extra Through measurement is required: Complement the minimum set of n-1 Throughs (as explained above) by an additional Through at a port pair that is connected by a "chain of throughs" of length 2 (e.g. between ports 2 and 4 in the examples above).

See Section 4.7.30, "External switch matrices", on page 271 for background information on switch matrices and their submatrices.

Compared to the full number of n(n-1)/2 Through connections, the time and effort is significantly reduced, in particular if n is large.

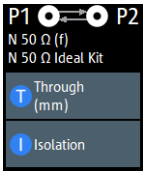
The "Reduced Through" logic is implemented for all full n-port calibration types.

- During manual calibration, you can apply the calibration when a sufficient set of Through connections have been measured. However, you can measure additional Through connections to improve the accuracy.
- For automatic calibrations with multiple port assignments, the calibration procedure always starts with a minimum set of port assignments, covering all ports and sharing a single common port to enable star-shaped Through measurements.
  - If "Use Reduced Number of Throughs for Cal Unit" on page 814 is enabled, the firmware applies the "Reduced Through" logic to each of these port assignments and does not propose additional calibration unit steps.
  - Otherwise, it measures all possible Throughs in each port assignment and offers additional port assignments to allow for a full n-port calibration ("Full Through").
    Once at least the minimum set of port assignments has been measured, the remaining ones can be skipped.



4.5.1.12 Complementary isolation measurement

For each port pair in a manual transmission normalization or TOSM calibration, the Through measurement can be complemented by an isolation measurement. This measurement accounts for possible crosstalk between the related test ports (e.g. on a test fixture).



If isolation is measured, the corrected transmission coefficient of the DUT is calculated as:

(Transmission coefficient DUT – Isolation) / (Transmission coefficient Through – Isolation)

There is no dedicated physical standard for isolation measurement; it is recommended to terminate the test ports suitably (e.g. with 50 Ω loads).

4.5.2 Calibration standards and calibration kits

A calibration kit is a set of physical calibration standards for a particular connector type. The magnitude and phase response of the calibration standards (i.e. their S-parameters) must be known or predictable within a given frequency range.

The standards are grouped into several types (Open, Through, Match,...) corresponding to the different input quantities for the analyzer's error models. The standard type also determines the equivalent circuit model used to describe its properties. The circuit model depends on several parameters that are stored in the cal kit file associated with the calibration kit.

As an alternative to using circuit models, it is possible to describe the standards with S-parameter tables stored in a file.

The analyzer provides many predefined cal kits but can also import cal kit files and create kits:

- A selection of predefined kits is available for all connector types. The parameters of these kits are displayed in the "View / Modify Cal Kit Standards" dialog, however, it is not possible to change or delete the kits.
- Imported and user-defined kits can be changed in the "Calibration Kits" dialog and its various subdialogs.

Calibration kits and connector types are global resources; the parameters are stored independently and are available irrespective of the current recall set.

4.5.2.1 Calibration standard types

The following table gives an overview of the different standards and their circuit models (offset and load models).

Table 4-8: Calibration standard types

Standard Type	Characteristics	Ideal Standard	Offset Model	Load Model
Open	Open circuit (one-port)	∞ Ω	☑	☑
Short	Short circuit (one-port)	0 Ω	☑	☑
Offset short	Short circuit with added electrical length offset, for waveguide calibration (one-port)	0 Ω	☑	☑
Match	Matched broadband termination (one-port)	Z_0 (characteristic impedance of the connector type)	☑	☑
Sliding match	One-port standard consisting of an air line with a movable, low-reflection load element (sliding load)	–	–	–
Reflect	Unknown mismatched standard (one-port)	∞ Ω	☑	☑
Through	Through-connection with minimum loss (two-port)	–	☑	–
Line1, Line 2	Line standards for TRL calibration with minimum loss (two-port)	–	☑	–
Attenuation	Fully matched standard in both directions (two-port; the reflection factor at both ports is zero)	–	–	–
Symm. network	Unknown mismatched reflection-symmetric standard (two-port)	–	☑	☑

Offset parameters

The offset parameters have the following physical meaning:

- The *delay* is the propagation time of a wave traveling through the standard. The *electrical length* is equal to the delay times the speed of light in the vacuum. It is a measure for the length of transmission line between the standard and the actual calibration plane. For a waveguide with permittivity ε_r and mechanical length L_{mech}, the following relations hold:

$$\text{Delay} = \frac{L_{mech} \cdot \sqrt{\varepsilon_r}}{c}; \quad \text{Electrical Length} = L_{mech} \cdot \sqrt{\varepsilon_r}$$

The default delay is 0 s, the default step width is 1 ns, corresponding to a step width of 299.792 mm for the electrical length. The relations hold for 1-port and 2-port standards.

- Z_0 is the characteristic impedance of the standard. If the standard is terminated with Z_0, then its input impedance is also equal to Z_0. Z_0 is not necessarily equal to the reference impedance of the system (depending on the connector type) or the terminal impedance of the standard. The characteristic impedance of the standard is only used in the context of calibration.
  The default characteristic impedance is equal to the reference impedance of the system.
- The *loss* is the energy loss along the transmission line due to the skin effect. For resistive lines and at RF frequencies, the loss is approximately proportional to the square root of the frequency.
  In Keysight mode, the *offset loss* is expressed in units of Ω/s at a frequency of 1 GHz. The following formula holds for two-port standards:

$$Offset\ Loss\,/\,[\Omega/s] = \frac{Loss\,/\,[dB]\cdot Z_0\,/\,[\Omega]}{4.3429\,/\,[dB]\cdot delay\,/\,[s]}$$

  The conversion formula for one-port standards has an additional factor ½ on the right-hand side. The reason for this factor is that the Loss in dB accounts for the attenuation along the forward **and** the reverse path. It does not depend on how often the wave actually propagates through the line, whereas the offset loss is proportional to the attenuation of the line.
  To determine an offset loss value experimentally, measure the delay in seconds and the loss in dB at 1 GHz and use the formula above.
  The default loss or offset loss is zero.

The impedance for waveguides is frequency-dependent. If a waveguide line type is selected in the "Cal Connector Types" dialog, the "Char. Imp." field is disabled and indicates "varies" instead of a definite impedance value. Moreover no loss or offset loss can be set.

Offset parameters and standard types

Offset parameters are used to describe all types of standards except the Sliding Match and the Attenuation.

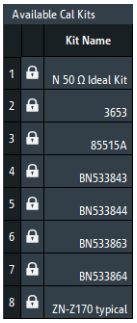
- The Sliding Match is a one-port standard with variable load parameters (sliding load) and unspecified length. The reference impedance is fixed and equal to the characteristic impedance of the connector type. No load and offset parameters need to be set.
- The Attenuation is a two-port standard which is fully matched in both directions (the reflection factor at both ports is zero). No load and offset parameters need to be set.

Load parameters and standard types

Load parameters are used to describe all types of standards except a Through, a Sliding Match, a Line, and an Attenuation.

- The Through standard is a through-connection between two ports with minimum loss which is taken into account by the offset parameters.
- The Sliding Match is a one-port standard with variable load parameters (sliding load), so there is no fixed load model.
- The Line standard is a line of variable length with minimum loss which is taken into account by the offset parameters.
- The Attenuation is a two-port standard which is fully matched in both directions (the reflection factor at both ports is zero). No load and offset parameters need to be set.

4.5.2.2 Cal kit parameter types



The analyzer uses three types of parameters to describe the calibration standards. The parameter type is the same for all standards in a kit and therefore appended to the kit name:

- **Universal** parameters (no suffix) describe calibration kit models with highly standardized components so that the parameters are valid for all calibration kits of the model.
- **Typical** parameters (suffix "typical") approximately describe a calibration kit model. To correct for deviations between the standards, each kit of the model is individually measured and delivered with an additional, kit-specific parameter set. Therefore each typical parameter set "<kit_name> typical" is complemented by an additional parameter set "<kit_name>" containing optimized parameters for an individual kit.
- **Ideal** parameters (suffix "Ideal Kit") describe an idealized calibration kit for each connector type; see below.

Make sure to use universal or individual parameter sets if you need to obtain high-precision results. The precision of the calibration kit parameters determines the accuracy of the system error correction and of the measurements. The R&S ZNB displays a warning if you use a typical or ideal parameter set to calibrate a channel.

Calibration kits can be obtained as network analyzer accessories; refer to the data sheet for the relevant ordering information. The name of all parameter sets is equal to the name of the corresponding calibration kit model.

Ideal parameters

All ideal kits contain the standards listed below.

Table 4-9: Ideal standard parameters

Standard (Gender)	R (Load)	Electrical Length (Offset)
Open (f, m)	∞ Ω	0 mm (Delay: 0 s)
Short (f, m)	0 Ω	0 mm
Offset Short (f, m)	0 Ω	10 mm

Standard (Gender)	R (Load)	Electrical Length (Offset)
Match (f, m)	Z_0 (characteristic impedance of the connector type)	0 mm
Sliding Match (f, m)	–	0 mm
Reflect (f, m)	∞ Ω	0 mm
Through (ff, mm, mf)	–	0 mm
Line (ff, mm, mf)	–	10 mm
Attenuation (ff, mm , mf)	–	0 mm
Symm. Network (ff, mm, mf)	–	0 mm

The following additional parameters are used:

- Characteristic impedance: Z_0 (characteristic impedance of the connector type)
- Loss: 0 dB / sqrt(GHz) or (0 GΩ / s) in Keysight mode
- All inductance and capacitance parameters are set to zero.

4.5.2.3 Sliding Match standards

The Sliding Match is a one-port standard consisting of an air line with a movable, low-reflection load element (sliding load). This standard is used because a no perfect Match is available over a wide frequency range. However, a series of measurements at a given frequency with equal mismatch and varying phase yields reflection factors that are located on a circle in the Smith chart. The center of this circle corresponds to perfect match. The network analyzer determines and further corrects this match point following I. Kása's circle-fitting algorithm.

To obtain the reflection coefficient for a perfectly matched calibration standard, the Sliding Match must be measured at least at 3 positions which should be unequally spaced to avoid overlapping data points. Increasing the number of positions to 4 to 6 can improve the accuracy. Moreover, using the predefined load positions of the standard is recommended.

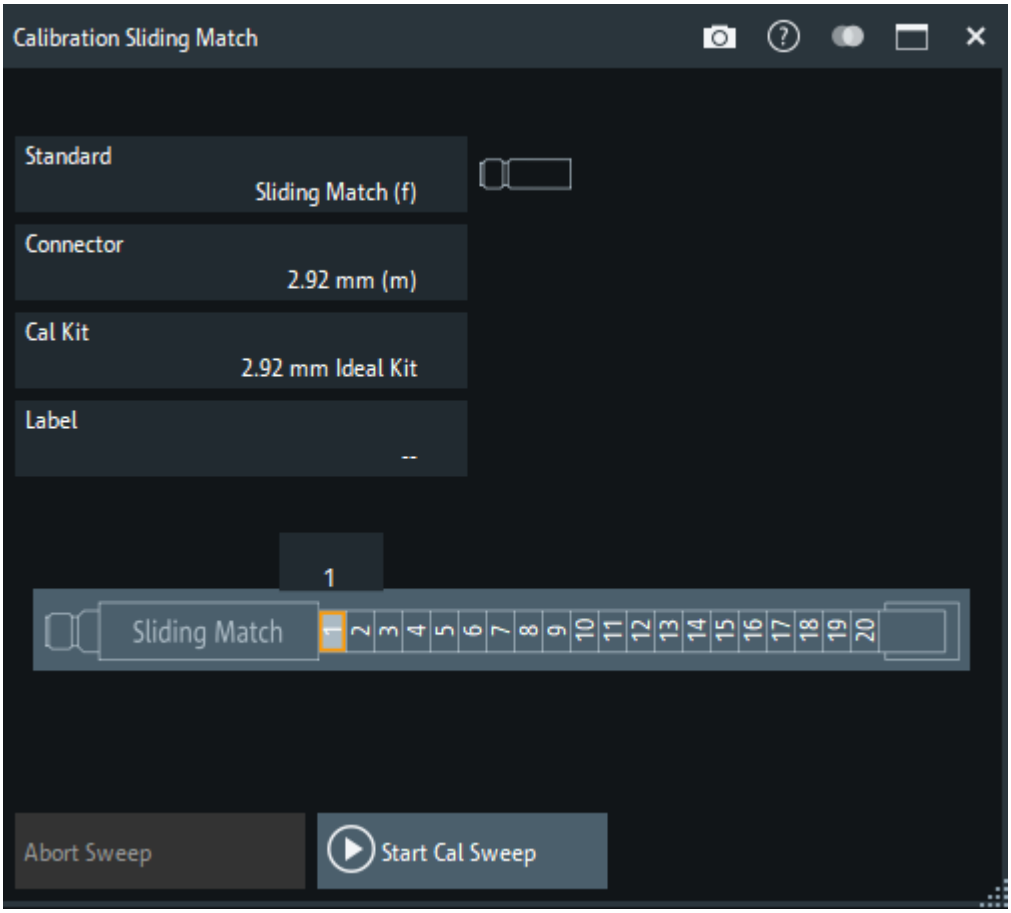


Figure 4-10: Sliding Match: GUI representation

A calibration is valid (and can be applied to the calibrated channel) if either the Match or three positions of the Sliding Match have been measured. However, it is often desirable to acquire calibration data from both standards.

The R&S ZNB can acquire correction data for up to 20 positions.

The analyzer combines the data in an appropriate manner:

- The Match results are used up to the lower edge of the specified frequency range of the Sliding Match (Min Freq).
- The Sliding Match results are used for frequencies above the Min Freq. In general, the Sliding Match provides better results than the Match within its specified frequency range.

4.5.2.4 Cal kit Files

Calibration kit files can be used to store the parameters of a particular calibration kit, to reload the data and to exchange calibration kits from one network analyzer to another.

Cal kit file contents

Cal kit files are independent of the current recall set and contain the following information:

- Name and label of the calibration kit
- Connector type including all connector type parameters (name, polarity, offset model, reference impedance)
- Type, gender and label of all standards in the kit together with the circuit model parameters (offsets, load) or S-parameter tables (.s<n>p file) that are necessary to determine its magnitude and phase response.

By default cal kit files are stored in the `C:\Users\Public\Documents\Rohde-Schwarz\Vna\Calibration` directory.

- To **export** cal kit data, the analyzer uses a specific binary file format `*.calkit`.
- Three different **import** file formats are supported: R&S ZVA-compatible binary cal kit files (`*.calkit`), R&S ZVR-specific binary cal kit files (`*.ck`), cal kit files in Keysight-specific ASCII format (`*.csv`).

Importing older R&S ZVR cal kit files

On loading some older R&S ZVR-specific `*.ck` files, e.g. the R&S ZV-Z23 cal kit file, the R&S ZNB generates the message "File does not comply with instrument calibration kit file format". The files must be converted using an R&S ZVR network analyzer equipped with a firmware version V3.52 or later.

Proceed as follows:

- On the R&S ZVR, press "CAL > CAL KITS > MODIFY KITS > INSTALL NEW KIT" to import the `*.ck` file.
- Press "CREATE INST FILE" in the same submenu to export the `*.ck` file in a R&S ZNB-compatible format.
- Import the converted file into the R&S ZNB.

Keysight *.csv cal kit files

The network analyzer can import and process cal kit files created with Keysight's VNA Cal Kit Manager (extension `*.csv`)

The import function expects a dot as decimal separator – otherwise it displays an error message.

4.5.3 Calibration pool

The calibration "Pool" is a collection of correction data sets (cal groups) that the analyzer stores in a common directory `C:\Users\Public\Documents\Rohde-Schwarz\Vna\Calibration\Data`. Cal groups in the pool can be applied to different channels and recall sets. Each cal group

is stored in a separate file named <CalGroup_name>.cal. The cal group name can be changed in the "Calibration Manager" dialog.

One of the available cal groups can be set as "Preset User Cal", i.e. the user correction data that should be restored after a user-defined preset.

If a new channel is created, the channel calibration of the active channel is also applied to the new channel. See also Calibration state labels.

4.5.4 Calibration state labels

The following labels in the trace list inform you about the status or type of the current system error correction.

Table 4-10: Calibration state labels (system error correction)

Label	Meaning
Cal	The system error correction is applied without interpolation. This means that a set of measured correction data is available at each sweep point.
Cal int	The system error correction is applied, however, the correction data for at least one sweep point is interpolated from the measured values. This means that the channel settings have been changed so that a current sweep point is different from the calibrated sweep points. It is not possible to disable interpolation.
Cal Off	The system error correction is no longer applied (e.g. "User Cal Active" is disabled). See also "Cal Off label" on page 155.

Table 4-11: Noise figure calibration state labels

Label	Meaning
NCal	A consistent set of power and noise figure calibration data is available at each sweep point. The calibration is applied.
NCal Off	The system error correction is no longer applied (e.g. "User Cal Active" is disabled). See also "Cal Off label" on page 155.

4.5.5 Automatic calibration

A calibration unit is an integrated solution for automatic system error correction of vector network analyzers. Rohde & Schwarz offers a wide range of calibration units for different frequency ranges and connector types. The connector types of the calibration unit should be selected according to the connector types of the DUT.

The calibration units contain calibration standards that are electronically switched when a calibration is performed. The calibration kit data for the internal standards is also stored in the calibration unit, so that the analyzer can calculate the error terms and apply the calibration without any further input.



Advantages of automatic calibration

Automatic calibration is faster and more secure than manual calibration, because:

- There is no need to connect several standards manually. The number of connections to be performed quickly increases with the number of ports.
- Invalid calibrations due to operator errors (e.g. wrong standards or improper connections) are almost excluded.
- No need to handle calibration kit data.
- The internal standards do not wear out because they are switched electronically.

4.5.5.1 Connecting the calibration unit

The calibration units provide the following connectors:

- USB type B connector at the rear, which is used to power-supply and control the unit. A USB cable for connection to the network analyzer is provided with the calibration unit.
- RF connectors, which are connected to the test ports. For all Rohde & Schwarz calibration units except a customized R&S ZN-Z51, the connector types are equal for all ports.

To connect the unit:

1. Switch on and power up your network analyzer.
2. To protect your equipment against ESD damage, use the wrist strap and grounding cord supplied with the instrument and connect yourself to the GND connector at the front panel.
3. Connect the USB type A connector of the USB cable to any of the USB type A connectors of the analyzer. Connect the USB type B connector of the USB cable to the USB type B connector of the calibration unit.
4. Wait until the operating system has recognized and initialized the new hardware. When the unit is connected for the first time, this may take longer than in normal use.

The unit is ready to be used, see Section 4.5.5.2, "Performing an automatic calibration", on page 177.



Recommended use

- The calibration unit is intended for direct connection to R&S ZNB network analyzers following the procedure described above. You can also connect the unit before switching on the analyzer. Do not connect the unit to other USB hosts, e.g. a PC, or insert any USB hubs between the analyzer and the unit, as this may damage the unit or the host.
- You can connect several calibration units to the different USB ports of the analyzer. You can also connect cal units and other devices (mouse, USB memory stick etc.) simultaneously.
- An unused calibration unit may remain connected to the USB port while the network analyzer is performing measurements. It must be disconnected during a firmware update.
- It is safe to connect or disconnect the calibration unit while the network analyzer is operating. Never connect or disconnect the unit while data is being transferred between the analyzer and the unit. Never connect the unit during a firmware update.

4.5.5.2 Performing an automatic calibration

After connection and initialization of the calibration unit, perform the automatic calibration of the related test ports using the "Calibration Unit" wizard (Channel – [Cal] > "Start Cal" > "Start (Cal Unit)"; see Section 5.11.1.3, "Calibration Unit wizard", on page 565). The wizard indicates the required port (re-)connections.



The assignment between the analyzer ports and the cal unit ports can be detected automatically. If auto-detection fails (e.g. because of a high attenuation in the signal path), you can either enter the port assignment manually or connect matching port numbers and select "Set to Default Port Assignment".

When finished, remove the test cables from the unit, connect your DUT instead and perform calibrated measurements.

Accuracy considerations

To ensure an accurate calibration, please observe the following items:

- Unused ports of the calibration unit must be terminated with a 50 Ω match.
- No adapters must be inserted between the calibration unit and the test ports of the analyzer.
- Allow for a sufficient warm-up time before starting the calibration. Refer to the specifications of the calibration unit for details.
- To ensure best accuracy, the analyzer automatically reduces the source power to -10 dBm. If the test setup contains a large attenuation, deactivate "Auto Power Reduction for Cal Unit" in the "Calibration" tab of the "System Config" dialog. Ensure an input power of -10 dBm at the ports of the calibration unit (please also refer to the specifications of the calibration unit).

NOTICE

Maximum RF input power

The maximum RF input power of the calibration unit is beyond the RF output power range of the analyzer, so there is no risk of damage if the device is directly connected to the test ports. If you use an external power amplifier, make sure that the maximum RF input power of the calibration unit quoted in the data sheet is never exceeded.

The available calibration types depend on the number of ports to be calibrated. For a single calibrated port, the reflection calibration types are available ("Refl Norm Open", "Refl Norm Short", "Refl OSM").

For n>1 ports to be calibrated, the analyzer provides the following additional calibration types:

- A full n-port (TOSM or UOSM) calibration for n calibrated ports.
- n full one-port calibrations.
- (n – 1) one path two port calibrations for n calibrated ports (all possible 2-port combinations from the "Node Port" to any other port). The node port is the source port for each one path two port calibration (fully calibrated port).
- (n – 1) transmission normalizations (bidirectional, forward or reverse) for n calibrated ports (all possible 2-port combinations from the first port to any other port). "Forward" transmission normalization means that the signal direction is from the ports with the lower numbers to the port with the higher numbers.

Example:

Select ports no. 1, 2, 3. A forward transmission normalization calibrates the S-parameters S_{21}, S_{31}, and S_{32}. A reverse transmission normalization calibrates the S-parameters S_{12}, S_{13}, and S_{23}. A bidirectional transmission normalization calibrates all six transmission S-parameters.

4.5.5.3 Characterization of calibration units

Each calibration unit is delivered with factory characterization data which ensure an accurate calibration for all standard applications. For specific modifications of the test setup, e.g. the connection of additional adapters to a calibration unit, a modified set of characterization data (suitable for the cal unit with adapters) may be desirable. The R&S ZNB provides a characterization wizard which you can use to generate your own characterization data sets for (modified) R&S cal units. The characterization data can be stored in the cal unit and used for automatic calibration whenever needed.

A cal unit characterization can be performed in a frequency sweep. The network analyzer must be properly calibrated, with the reference plane at the input ports of the (modified) cal unit to be characterized.

The procedure involves the following steps:

1. Perform a calibration of your network analyzer, using the test setup and the calibration type you wish to perform with your calibration unit.
2. Connect the calibration unit to the network analyzer.
3. Access the "Characterize Cal Unit" dialog (Channel – [Cal] > "Cal Devices" > "Characterize Cal Unit") and select "Start Characterization".
4. Step through the "Characterization" wizard, following the instructions in the dialogs.

Dependency between calibration types and characterization data

A cal unit characterization provides full one-port (OSM) data at the selected ports plus two-port (Through) data between any pair of selected ports. The measurement of Through data is optional, however, it is required for some calibration types. The following table gives an overview.

Calibration type	Characterization data required
Refl Norm Open Refl Norm Short Refl OSM UOSM TOSM	OSM CalPort 1, OSM CalPort2 ... (all calibrated ports)
Trans Norm Both Trans Norm Forward One Path Two Ports	OSM CalPort 1, OSM CalPort2 ... (all calibrated ports), Through (between all pairs of ports)

4.5.5.4 Inline calibration

Any disturbance of the measurement setup after the calibration process inevitably produces errors. If a massive system error correction is required, e.g. if long RF cables with high damping are used, those disturbances can lead to inaccurate and unreproducible measurement results. Especially in remote scenarios, such as measurements in a

thermal vacuum chamber (TVAC), a recalibration using traditional calibration kits or calibration units is a costly and time consuming process.

The Rohde & Schwarz inline calibration system (ICS) R&S ZN-Z3x allows recalibrating a running measurement setup without mechanically changing it. For each DUT port, an inline calibration unit (ICU) is inserted between the corresponding VNA test port and the DUT. The ICU remains within the signal path throughout the measurement (low-loss through path) and, on request of the VNA, performs a full one-port calibration (OSM) in-situ.

For n-port measurements with n>2, an initial n-port calibration of the test setup has to be done.



Figure 4-11: ICU R&S ZN-Z32

Control connections are established via a central Inline Calibration Controller (ICC) with:

- 2 CAN bus interfaces for connecting the ICUs
- USB and LAN interface for remote control (from the R&S ZNB or a standard Windows PC)

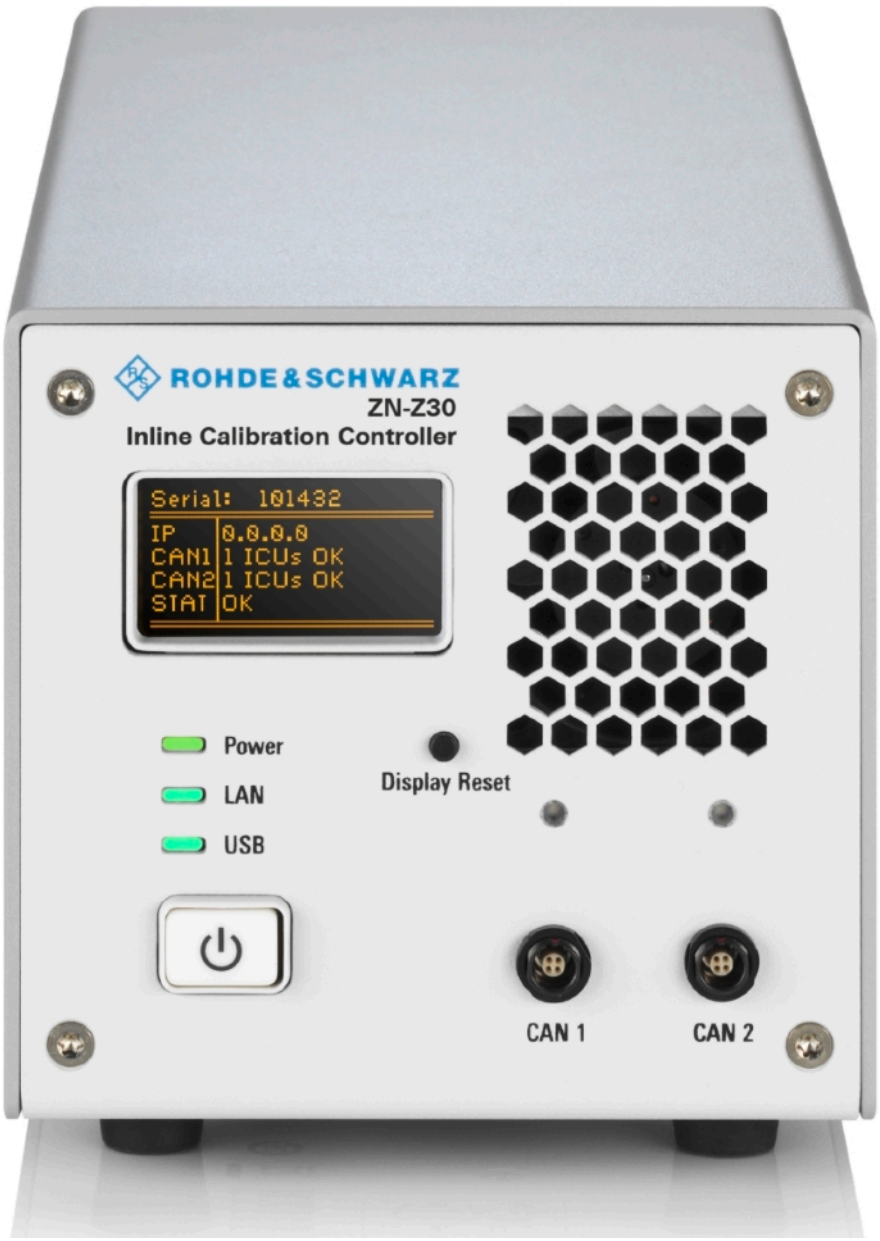


Figure 4-12: ICC R&S ZN-Z30

For more information, see the R&S ZN-Z3x product pages at https://www.rohde-schwarz.com/product/NetworkAnalyzer_Acc_ZNZ3.

Firmware integration

The control connection between R&S ZNB and ICC is established via USB. After the VNA firmware has detected the ICC, it requests information about connected ICUs from the ICC.

Because the ICC does not support hot plugging of ICUs, connect the ICUs to the ICC before connecting the ICC to the R&S ZNB.

The VNA firmware supports:

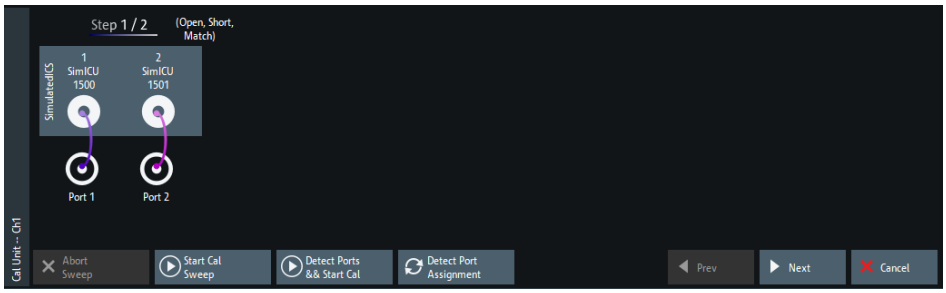
- Pulling characterization data for the connected ICUs from the ICC
- Interpolation and extrapolation of factory ICU characterization data to match the current ICU temperature
- Performing user characterizations of ICUs:
  - Embedding the Through standard of the ICU into its Open, Short, and Match standards
  - Storing the resulting characterization on the ICU
- Performing full n-port system calibrations using n ICUs
- Performing system (re-)calibrations

- The characterization and calibration functionality for standard calibration units is also used for ICUs.
- Embedding the Through into the ICU's OSM standards is necessary, because the ICU remains in situ. For this reason, during ICU characterizations the Through measurement is mandatory.

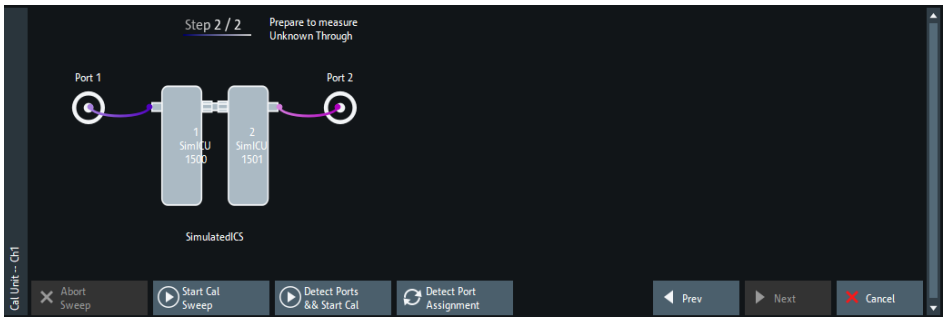
Workflow

1. Configure the ICS as required, connecting all the ICUs to the ICC via CAN bus.
2. Connect the ICC to the VNA via USB.
3. Perform user characterizations of the connected ICUs (optional).
   a) Attach the cabling/adapters that are required to connect both ports of an ICU to the VNA.
   b) Calibrate this setup using a manual calibration kit or a calibration unit – without connecting the ICUs
   c) Connect the ICUs and proceed with the user characterization.
4. Perform an initial (automatic) calibration of your test setup.
   a) Select the ICS as calibration unit.
   b) Assign the correct ICUs to the related ports.
   c) Select suitable characterization data for each ICU.
   d) For factory characterizations, activate temperature compensation (optional).
   e) Start the calibration.

f) Step 1: OSM



g) Step 2: Unknown Through



5. Connect the DUT to the calibrated setup and perform the required measurements.
6. Refresh the calibration in situ, whenever required.
   a) "Repeat" the calibration
   b) Perform OSM measurements.
   c) Skip Through measurements.

4.5.5.5 Calibration validation

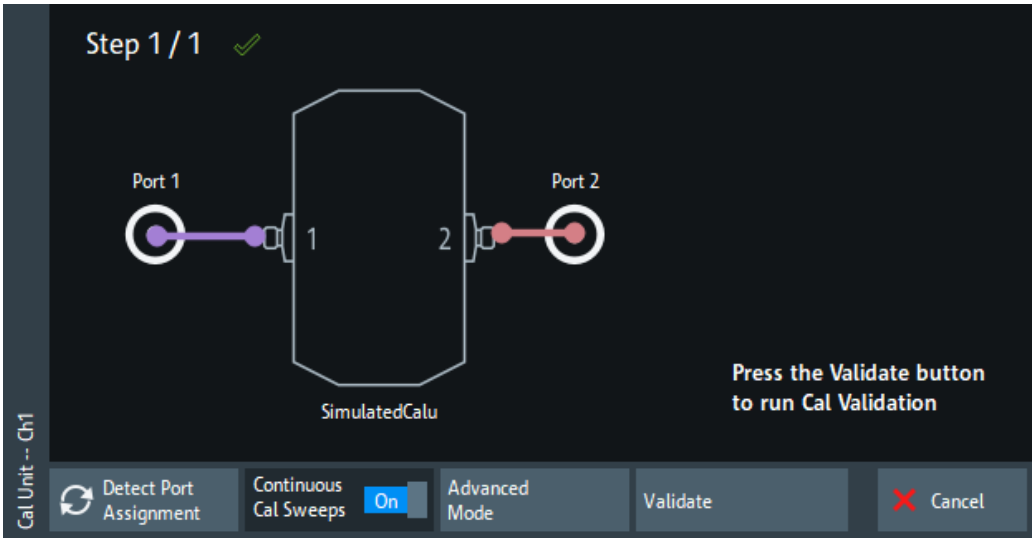
This feature allows you to validate the active channel calibration against the expected result from a stored characterization. It requires an R&S calibration unit.

- Cal validation works with at most 4 calibrated ports and with 2- and 4-port calibration units.
- Currently, cal validation is limited to automatic calibrations.

Calibration validation provides an automated, single-click (basic) mode, and an advanced mode for detailed validation and adjustment of limit tolerances, reporting, and SCPI automation.

Basic mode

When "Cal Validation" is launched for the first time, it runs in basic mode. In this mode, the R&S ZNB automatically detects the instrument ports connected to the cal unit, and only validates connected ports. Thus, if a 2-port cal unit is connected to a 4-port calibrated channel, only the two connected instrument ports are validated against the cal unit.



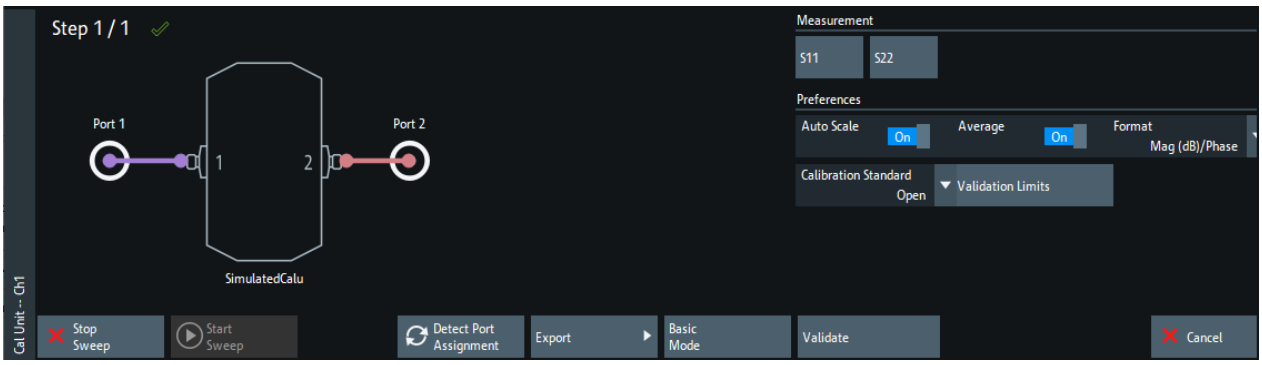
The validation logic proceeds through the connected ports p ∈ {1, ..., 4} in ascending order, measuring S_{pp} for the Open, Short and Match standard. It then compares the measured S_{pp} values with the corresponding reflection S parameters of the cal unit's factory characterization. The on-screen feedback indicates success ("PASS", failure ("FAIL") or provides a warning about undetected ports.



For a channel with more calibrated ports than cal unit ports available, you can change the port assignments and rerun the automated validation. Or you can switch to advanced mode and validate each port manually.

Advanced mode

In advanced mode you can configure the validation settings (reference characterization, deviation limits) and run or rerun particular measurements (reflection at port p with connected Open, Short or Match).



4.5.5.6 Multiple port assignments

With multiple port assignments, you can calibrate port groups of any size, no matter how many ports your calibration unit (or cal unit characterization) has. For example, you can use a 2-port calibration unit to perform a full 4-port calibration.

The multiple port assignment method considerably extends the range of applications of the calibration units. However, the method entails some loss of convenience because you have to reconnect the calibration unit between the different calibration stages (assignments). It can also cause a loss of accuracy because only a subset of all possible through connections is measured.

To calibrate n test ports with an m-port calibration unit (m<n), the calibration unit has to be connected at least n/m times. Each connection is described by its "port assignment", i.e. the mapping of calibration unit ports to test ports. Then for each assignment an automatic calibration is performed. Finally the analyzer combines the calibration data and calculates the required n-port error terms.

This calculation is possible if and only if:

- The port assignments "cover" the calibrated test ports
- The overlap between assignments allows a(n ordered) "chain of measured through connections" between any (ordered) pair of test ports requiring a Through measurement

For full n-port calibrations, to calculate the correction terms for those test port pairs that are not covered by a single assignment, the R&S ZNB applies the "reduced through" logic. See Section 4.5.1.11, "Full n-Port calibration with reduced number of Through connections", on page 166. Depending on the calibration type, a "minimal" valid and complete solution can be described as follows:

Calibration type	Minimal solution	Default solution (minimal)
Full One Port	Each calibrated test port must appear in exactly one port assignment.	Subdivide the n test ports into groups of m ports with increasing port numbers. Create a separate port assignment for each group.
One Path Two Port	• The node port must be included in all port assignments • Each of the n-1 other calibrated test ports must appear in exactly one port assignment *Additional condition:* The calibration unit port assigned to the node port must be the same in all assignments. This condition minimizes the number of port reconnections between the calibration stages.	Assign the node port to port 1 of the calibration unit. Subdivide the remaining n–1 test ports into groups of m–1 ports and create a separate port assignment for each group. Leave port 1 of the calibration unit connected to the node port and connect the remaining ports of each port group to the remaining ports of the calibration unit.
Full n-Port	• the two generic conditions stated above (with "unordered Throughs") • the number of assignments must be as small as possible *Additional condition:* The cal unit port assigned to a given test port must be the same in all assignments. Again, this condition minimizes the number of port reconnections between the calibration stages.	Same as for One Path Two Port calibrations, using the test port with the lowest number as "node port". (Every other calibrated port could serve as "node port")

Among the minimal ones, the "star-shaped" solutions with fixed (but arbitrary) "node port" are also optimal w.r.t. the (average) length of the "Chain of Throughs" and hence the accuracy of the calculated Throughs. Additional port assignments can improve the accuracy, if they add more measured through connections.

For full n-port calibrations, an extra assignment is required if:

- An external switch matrix is used
- You want to calibrate 3 or more test ports of the **same submatrix**
- Only 2 cal unit ports are available

Starting with the minimal solution (as explained above), create an additional 2-port assignment for a pair of ports that was previously not covered by a single, but by two assignments.

See Section 4.7.30, "External switch matrices", on page 271 for background information on switch matrices and their submatrices.

Example:

The following examples show minimal port assignments for a Full 9-Port calibration using a four-port calibration unit:

Table 4-12: Full 9-port calibrations: star-shaped optimum solution

Test Port	Assignment 1	Assignment 2	Assignment 3
1	Cal Unit Port 1	Cal Unit Port 1	Cal Unit Port 1
2	Cal Unit Port 2	-	-

Test Port	Assignment 1	Assignment 2	Assignment 3
3	Cal Unit Port 3	-	-
4	Cal Unit Port 4	-	-
5	-	Cal Unit Port 2	-
6	-	Cal Unit Port 3	-
7	-	Cal Unit Port 4	-
8	-	-	Cal Unit Port 2
9	-	-	Cal Unit Port 3

Table 4-13: Full 9-port calibrations: line-shaped optimum solution

Test Port	Assignment 1	Assignment 2	Assignment 3
1	Cal Unit Port 1	-	-
2	Cal Unit Port 2	-	-
3	Cal Unit Port 3	-	-
4	Cal Unit Port 4	Cal Unit Port 4	-
5	-	Cal Unit Port 1	-
6	-	Cal Unit Port 2	-
7	-	Cal Unit Port 3	Cal Unit Port 3
8	-	-	Cal Unit Port 1
9	-	-	Cal Unit Port 2

4.5.6 Scalar power calibration

The purpose of a scalar power calibration is to ensure accurate source power levels and power readings at a particular position (calibration plane) in the test setup. Scalar power calibration is different from the system error correction described in Section 4.5, "Calibration", on page 154.

A power calibration is required for accurate measurement of wave quantities or ratios (see section Section 4.1.5, "Data flow", on page 89). For best accuracy, choose a calibration method according to Section 4.5.8, "Recommended calibration methods", on page 196.

Calibration of S-parameters

S-parameters are not affected by a scalar power calibration. S-parameters are ratios of incident and outgoing waves: for linear DUTs, they do not depend on the absolute power. For measurements on nonlinear DUTs, a SMARTerCal is recommended.

A SMARTerCal is also appropriate for frequency conversion measurements. For detailed information, refer to Section 4.5.7, "SMARTerCal", on page 193.

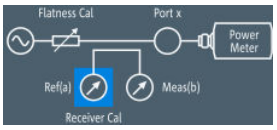
In general, a power calibration involves three stages:

1. Reference receiver calibration
   Connect an external power meter to the calibration plane and let the analyzer use power meter readings to calibrate its reference receiver.
2. Source flatness calibration
   The analyzer modifies its source power so that the reading of the calibrated reference receiver corresponds to the desired source power value.
3. Measurement receiver calibration
   The analyzer uses the calibrated source signal to adjust the power reading at the receive port.

4.5.6.1 Reference receiver calibration

The reference receiver calibration requires an external power meter, to be connected via GPIB bus, USB or LAN interface. The power sensor can be connected directly at the calibration plane or to any other point in the test setup where the signal power is known to be proportional to the power at the calibration plane.

The analyzer performs a first calibration sweep at the source power that is likely to produce the target power ("Cal Power") at the calibration plane. You can specify a known attenuation or gain at the source port and in the signal path between the source port and the calibration plane:



In the calibration sweep diagram, the analyzer displays:

- The power that the external power meter measures at the calibration plane
- The power measured at the reference receiver

It then uses the difference between the two traces to correct the reference receiver reading.

4.5.6.2 Source flatness calibration

A source flatness calibration ensures accurate power levels of the generated waves at an arbitrary calibration plane in the measurement path. Typically the calibration plane corresponds to the input of the DUT.

In a frequency sweep, the power at the calibration plane is maintained at a constant "Cal Power" value. The source power calibration eliminates frequency response errors in the signal path between the source and the calibration plane. It is possible to introduce an arbitrary attenuation or gain into the signal path so that the cal power is not restricted to the power range of the source. A typical application for a power calibration in a frequency sweep is the measurement of the gain of an amplifier across a frequency range but at a fixed input power.

In a power sweep, the power calibration ensures that the power at the calibration plane is either constant or a linear function of the stimulus power. A typical application for a power calibration in a power sweep is the measurement of the gain of an amplifier across a power range but at a fixed frequency. The correction data acquired in a frequency or power sweep is reused if a "Time" or "CW Mode" sweep is activated.

Calibration procedure

By default, the source flatness calibration involves several steps:

1. Reference receiver calibration
2. **Internal source power flatness calibration:** In the following steps, the firmware uses the calibrated reference receiver to adjust the source power. To this end, the VNA performs a series of calibration sweeps at varying source power until the number of "Total Readings" is reached or until the deviation between the calibrated reference receiver power and the cal power is below a specified "Tolerance". The external power meter is no longer used for these repeated calibration sweeps; everything is based on the previously calibrated reference receiver. This speeds up the calibration procedure but does not impair its accuracy.

   At the end of this step, the VNA performs an additional verification sweep to demonstrate the accuracy of the calibration.

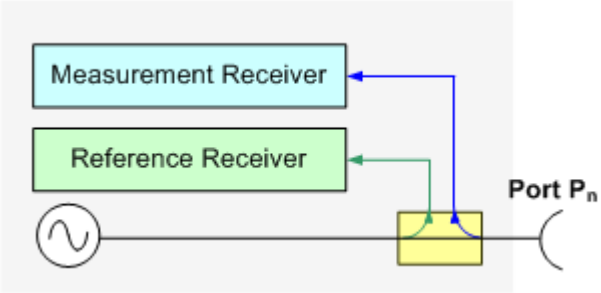
   Tip: It is also possible to perform the source flatness calibration using the power meter (without previously calibrating the reference receiver).
   This variant is particularly suitable for redefined physical ports with source and reference receiver assigned to different VNA ports.

After the source flatness calibration, you can expect the power at the calibration plane to be within the range of uncertainty of the power meter. The reference receiver reading corresponds to the calibrated source power. After a change of the sweep points or sweep range, the analyzer interpolates or extrapolates the calibration data; see Section 4.5.6.4, "Power calibration labels", on page 191.

4.5.6.3 Measurement receiver calibration

A measurement receiver calibration ensures that the power readings at a specified receive port of the analyzer (b-waves) agree with the source power level calibrated at an arbitrary calibration plane. Typically, the calibration plane is at the input of the receiver so that the calibration eliminates frequency response errors in the calibrated receiver.

In contrast, the reference receiver calibration ensures correct power readings for the generated waves (a-waves).



A measurement receiver calibration generally improves the accuracy of power (wave quantity) measurements. The correction data acquired in a frequency or power sweep is reused if a "Time" or "CW Mode" sweep is activated.

Calibration procedure

The measurement receiver calibration is based on a received wave b_n with known power. The calibration involves a connection to a (previously flatness-calibrated) source port.

The received wave to calibrate can be generated by a different analyzer port P_m ($m \neq n$) or by an external generator. Alternatively, it is possible to connect an Open or Short standard to port P_n: The measurement receiver is calibrated using the reflected wave a_n.

The measurement receiver calibration involves a single calibration sweep. The calibration sweep is performed with current channel settings but with a maximum IF bandwidth of 10 kHz. Smaller IF bandwidths are maintained during the calibration sweep; larger bandwidths are restored after the sweep. The analyzer measures the power at each sweep point, compares the result with the nominal power of the source, and compiles a correction table.

At the end of the calibration sweep, a checkmark symbol next to the calibrated source indicates the status of the measurement receiver calibration. After a change of the sweep points or sweep range, the analyzer interpolates or extrapolates the calibration data.

Calibration with non-ideal standards

A measurement receiver calibration typically assumes ideal transmission or reflection standards. For some applications, however, it is impractical or even impossible to connect the ideal standard directly to the ports or port, e.g. due to fixtures like a switch matrix or other electrical networks in between. With frequency-dependent losses in the fixture, assuming 100% reflection is wrong and the measurement receiver calibration is inaccurate.

To compensate for these losses during the calibration sweep, the R&S ZNB allows you to load correction data.

This functionality can only be used via SCPI. See:

- `SOURce<Ch>:POWer:CORRection:NISTandards:DEFine`
- `SOURce<Ch>:POWer:CORRection:NISTandards:LOAD`

- SOURce<Ch>:POWer:CORRection:NISTandards:DELete

4.5.6.4 Power calibration labels

Power calibration labels in the trace list for wave quantities and ratios inform you about the status and type of the current scalar power calibration. The labels appear in the following instances:

- For a-waves, if a Source flatness calibration is available.
- For b-waves, if a Measurement receiver calibration is available.
- For ratios between a- and b-waves, if both a source flatness and a measurement receiver calibration are available.

Calibration of S-parameters

S-parameters and derived quantities (e.g. impedances, admittances, stability factors) are assumed to be linear; otherwise they must be corrected by a SMARTerCal.

Therefore, a scalar power calibration is not applied to S-parameters and derived quantities; no power calibration labels appear in the trace list.

Table 4-14: Power calibration labels

Label	Meaning
PCal	A scalar power calibration is available and applied without interpolation or extrapolation (see below). This means that a set of measured correction data is available at each sweep point.
PCai	The power calibration is applied, however, the correction data for at least one sweep point is interpolated from the measured values. This means that the channel settings have been changed so that a current sweep point is different from the calibrated sweep points. It is not possible to disable interpolation.
PCao	The power calibration is applied, however, the source power (channel base power) is now set to a different level (**o**ffset) compared to the level that was used during calibration.
PCax	The power calibration is applied, however the calibration data is extrapolated. The current stimulus range exceeds the calibrated stimulus range. The power calibration data of the first calibrated sweep point is used for all smaller stimulus values; the power calibration data of the last calibrated sweep point is used for all larger stimulus values.
PCa<l|i|o|x> S	Like PCa<l|i|o|x>, but only a source flatness calibration is available
PCa<l|i|o|x> R	Like PCa<l|i|o|x>, but only a receiver calibration is available
PCal Off	The power calibration is no longer applied (e.g. deliberately turned off in the "Calibration > Use Cal" softtool panel).

A lower label in the list has priority over the higher labels (e.g. if the power calibration is interpolated and the source power is changed, then the label PCao is displayed).

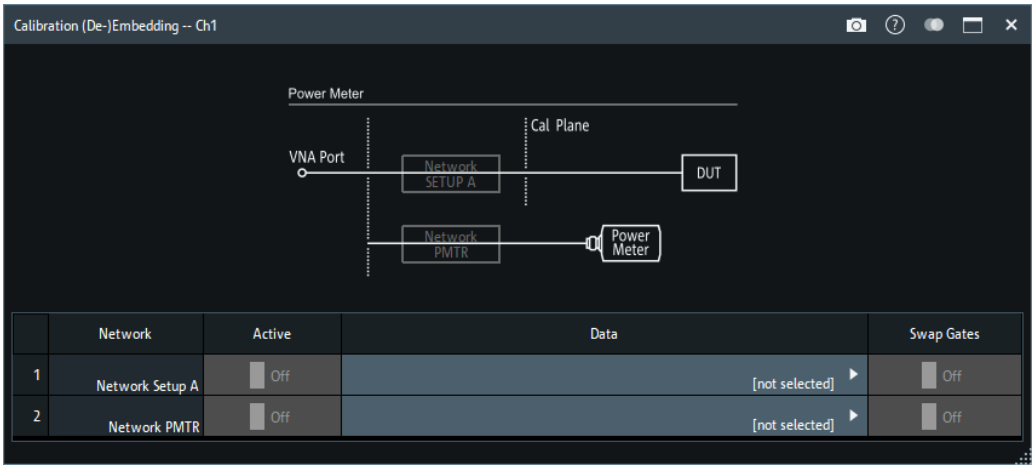
Interpolation and extrapolation

The analyzer can interpolate and extrapolate power correction data so that a source or receiver power calibration can be reused after a change of the frequency sweep range:

- At new sweep points within the calibrated sweep range, interpolation is applied to calculate the correction data. A label "PCai" in the trace list indicates an interpolated power calibration.
- At new sweep points outside the calibrated sweep range, the correction values are extrapolated: Sweep points below the lowest calibrated frequency are assigned the correction value of the lowest frequency. Sweep points above the highest calibrated frequency are assigned the correction value of the highest frequency. A label "PCax" in the trace list indicates an extrapolated power calibration.

4.5.6.5 Extended test setups

The power calibration data can be modified to account for an additional two-port device in the test setup. The known transmission coefficients of the two-port can be entered manually or loaded from a trace of the active VNA setup ([Cal] > "Power Cal Settings" > "Calibration (De-)Embedding").



The R&S ZNB supports two extensions.

A: Two-port at DUT (during measurement)

The upper connection variant in the figure above.

Test and measurement procedure:

1. Perform the calibration without the additional two-port. During the calibration, the analyzer decreases the power sensor values by the 2-port transmission coefficients to move the calibration plane of the power calibration towards the input of the DUT. The calibration plane corresponds to the output of the 2-port that is placed in-between the network analyzer port and the DUT.
2. Perform the measurement with the additional two-port.

In a **one-port setup**, to calibrate the measurement receiver using the calibrated reference receiver, you must connect a Short standard to the DUT side of the 2-port, and not to the VNA port.

B: Two-port at power meter (during calibration)

Test and measurement procedure:

1. Perform the calibration with the additional two-port between the analyzer port and the power sensor. During the calibration, the analyzer increases the power sensor values by the 2-port transmission coefficients to move the calibration plane of the power calibration towards the input of the DUT. The calibration plane corresponds to the input of the additional 2-port.
2. Perform the measurement without the additional two-port.

Practical example: An adapter or attenuator with known attenuation is needed to connect the power sensor to the test port of the network analyzer. The transmission coefficients of the adapter are used for the power meter correction.

4.5.7 SMARTerCal

SMARTerCal (smarter calibration) is a combination of a system error correction with a (scalar) receiver power calibration. The system error correction can be done for "One Path Two Ports" or as a full n-port system error correction (TOSM, UOSM, Adapter Removal, TRL, TNA ...) for two or more ports. The power calibration is carried out for a single port. If the full n-port system error correction has been applied, the results can be used to extend the one port calibration to the other ports. This power calibration extension is automatically done.

The calibration methods serve different purposes:

- The system error correction requires a set of calibration standards; it provides vector error-corrected S-parameters. For equal port frequencies, the n-port calibration types provide the full set of error terms. For frequency conversion measurements, a source match correction and (optional) load match correction is calculated.
- The receiver power calibration requires an external power meter; it corrects the power readings of the reference and measurement receivers according to the measured absolute power at the calibration plane. This does **not** include a readjustment of the actual source power (flatness calibration).
  Example: Channel base power: –10 dBm; the test setup involves a 3-dB attenuation between the source port and the calibration plane. After the power calibration is applied, the analyzer indicates an output power (a-wave) of –13 dBm, although the actual source power remains at –10 dBm.

The SMARTerCal is also applied to ratios and wave quantities. For measurements on linear DUTs, SMARTerCal is sufficient. Non-linear measurements can be further improved by a combination of a SMARTerCal plus a scalar power calibration. See Combining SMARTerCal with scalar power calibration.

For an overview of measurements and recommended calibration methods refer to Table 4-17.

4.5.7.1 Calibration procedure

A SMARTerCal is a fully menu-guided process which is performed like a regular system error correction. The calibration wizard defines the calibrated ports and the calibration type; it also initiates the calibration sweeps for all calibration standards. The calibration sweep for the external power meter is performed in analogy to a sweep for a one-port calibration standard. However, the analyzer uses the "Reference Receiver Cal Power" setting from the scalar power calibration. The order of the system error correction and power calibration sweeps is arbitrary; ensure that you always connect the proper equipment.

The R&S ZNB also supports a SMARTerCal based on the calibration units. The calibration units provide the n-port system error correction data (TOSM or UOSM); a subsequent power calibration sweep completes the calibration.

4.5.7.2 Calibration types

The names of the SMARTerCal calibration types consist of a prefix "P" (indicating the additional power calibration) plus the system error correction type.

The R&S ZNB supports the SMARTerCal equivalent of all full n-port system error corrections; an overview is shown below.

Table 4-15: SMARTerCal calibration types

SMARTerCal Type	Based on ...	Manual calibration	Calibration unit
P-One Path Two Ports	One Path Two Ports	☑	☑
PTOSM	TOSM	☑	☑
PUOSM	UOSM	☑	☑
PTRL	TRL	☑	☐
PTOM	TOM	☑	☐
PTSM	TSM	☑	☐
PTRM	TRM	☑	☐
PTNA	TNA	☑	☐
P-Adapter Removal	Adapter Removal	☑	☐

The selection criteria for the SMARTerCal calibration types are identical to the criteria for system error corrections. For an overview, refer to Section 4.5.1, "Calibration types", on page 155.

4.5.7.3 Combining SMARTerCal with scalar power calibration

As described in Section 4.5.7, "SMARTerCal", on page 193, the power meter results of a SMARTerCal are only used to calibrate the receivers of the R&S ZNB. To obtain definite source power levels, you can combine the SMARTerCal with an additional scalar source power calibration.

The scalar source power calibration and the SMARTerCal can be performed in any order. As a result of the combined calibration, the R&S ZNB measures vector error-corrected S-parameters; the scalar power correction is applied to wave quantities and ratios. Notice that this provides different values, e.g., of S_{21} and the ratio b_2/a_1. The flatness calibration step of the scalar source calibration ensures accurate input powers at the DUT.

A combined SMARTerCal and scalar power calibration is also appropriate for frequency conversion measurements on non-linear DUTs. For linear measurements, where the actual input power at the DUT is not critical, a SMARTerCal is sufficient.

4.5.7.4 Calibration labels

The status and type of the SMARTerCal is indicated in the trace list, in analogy to a system error correction (see Section 4.5.4, "Calibration state labels", on page 175). The calibration labels for a SMARTerCal consist of a prefix "S" plus the system error correction labels.

Table 4-16: Calibration labels (SMARTerCal)

Label	Meaning
SCal	The SMARTer Cal is applied without interpolation. This means that a set of measured correction data is available at each sweep point.
SCal int	The SMARTer Cal is applied, however, the correction data for at least one sweep point is interpolated from the measured values. This means that the channel settings have been changed so that a current sweep point is different from the calibrated sweep points. It is not possible to disable interpolation.
Cal Off	The SMARTer Cal is no longer applied (e.g. "User Cal Active" is disabled). See also "Cal Off label" on page 155.

The receiver power calibration included in the SMARTerCal is not indicated separately. If a SMARTerCal is combined with an additional scalar source power calibration (see Section 4.5.7.3, "Combining SMARTerCal with scalar power calibration", on page 195), the trace list may contain an addition power calibration label according to Section 4.5.6.4, "Power calibration labels", on page 191.



4.5.8 Recommended calibration methods

Table 4-17: Recommended calibration methods for various measurements

Measurement	System error correction	Scalar Power calibration	SMARTerCal[1)]
S-parameter meas. on linear DUTs	**Yes**	Not necessary	Not necessary
S-parameter meas. on nonlinear DUTs	Not possible --> Use SMARTerCal	**Yes** Power (source) and meas. receiver, supported in combination with SMARTerCal	**Yes**
Meas. of wave quantities or ratios on linear DUTs	Not possible --> Use SMARTerCal	Not necessary	**Yes**
Meas. of wave quantities or ratios on nonlinear DUTs	Not possible --> Use SMARTerCal	**Yes** Power (source) and meas. receiver, supported in combination with SMARTerCal	**Yes**
Power sweep, e.g. for compression point measurement	**Yes**	**Power (source): Yes** Meas. receiver: not necessary	Not necessary
Frequency conversion measurements on linear DUTs	Not possible --> Use SMARTerCal	Not necessary	**Yes**
Frequency conversion measurements on nonlinear DUTs	Not possible --> Use SMARTerCal	**Yes** Power (source) and meas. receiver, supported in combination with SMARTerCal	**Yes**

[1)] SMARTerCal is a multiport calibration type. For one-port measurements, perform a two-port SMARTerCal for a port pair including the measurement port.

4.5.9 Parallel calibration of multiple channels

If multiple channels are configured in the active recall set, clearly they can be calibrated one after the other, but this can be inefficient in terms of necessary reconnections of calibration standards (or calibration units).

The R&S ZNB offers multiple possibilities to calibrate several channels in parallel:

- Calibrate multiple channels in one go, using the same calibration type on the same ports for all channels
  In this case, for each port to be calibrated the same calibration standards have to be connected. After connecting one of these standards, a calibration sweep has to be performed for each channel.
  This simple mode of parallel calibration is supported from the analyzer GUI ("Calibrate all Channels" control in all calibration wizards) and via remote control (see `[SENSe:]CORRection:COLLect:CHANnels:ALL` on page 1355).
- Calibrate a subset of the available channels, possibly using different ports and calibration types.
  In this case, for each port to be calibrated a different set of calibration standards might be required. For each of these standards, only a subset of the available channels might have to be swept. See `[SENSe:]CORRection:COLLect:CHANnels:MCTYpes` on page 1355.

To its full extent, this advanced mode of parallel calibration is available via **remote control only**.

4.5.10 Joining calibrations

The "JoinCal" functionality of the analyzer firmware allows you to join two calibrations from an active channel or the calibration pool. The resulting calibration can replace one of the joined calibrations, can be stored in a new channel, or added to the calibration pool.

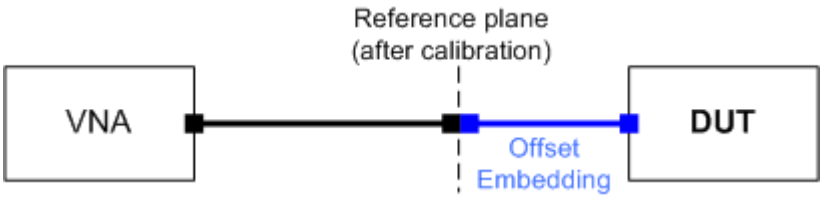
- **Frequency range join**
  Joins calibrations of the same port set, but with different overlapping frequency ranges. The resulting calibration is of the same kind and applies to the same port set, but covers the joint frequency range.
- **Port set join**
  Joins calibrations with the same frequency range, but with different port sets. The resulting calibration is of the same kind and covers the same frequency range, but applies to the joint port set
  Calibrations on additional frequency axes in the *second* input calibration are copied to the output calibration.

4.6 Offset parameters and de-/embedding

The R&S ZNB functionality described in this section complements the calibration, compensating for the effect of known transmission lines or matching networks between the calibrated reference plane and the DUT.

4.6.1 Offset parameters

Offset parameters compensate for the known length and loss of a (non-dispersive and perfectly matched) transmission line between the calibrated reference plane and the DUT.



The analyzer can also auto-determine length and loss parameters, assuming that the actual values should minimize the group delay and loss across a configurable frequency range.

4.6.1.1 Definition of offset parameters

The *delay* is the propagation time of a wave traveling through the transmission line. The *electrical length* is equal to the delay times the speed of light in the vacuum. It is a measure for the length of the transmission line between the standard and the actual calibration plane. For a line with permittivity ε_r and *mechanical length* L_{mech} the delay and the electrical length are calculated as follows:

$$Delay = \frac{L_{mech} \cdot \sqrt{\varepsilon_r}}{c};\ Electrical\ Length = L_{mech} \cdot \sqrt{\varepsilon_r}$$

In the CHANNEL – [OFFSET EMBED] > "Offset" softtool tab, "Delay","Electrical Length" and "Mech. Length" are coupled parameters. When one of them is changed, the other two are adjusted accordingly.

For a non-dispersive DUT, the delay defined above is constant over the considered frequency range and equal to the negative derivative of the phase response for the frequency (see mathematical relations). The length offset parameters compensate for a constant delay, which is equivalent to a linear phase response.

4.6.1.2 Definition of loss parameters

The loss is the attenuation a wave experiences when traveling through the transmission line. In logarithmic representation, it can be modeled as the sum of a constant and a frequency-dependent part, where the frequency dependence is due to the skin effect.

In the limit case, where the length of the transmission line is considered to be "almost zero", the loss is considered constant:

$$Loss(f) = Loss_{DC}$$

Otherwise, if the loss at DC and one additional frequency f_1 is known (or measured), the loss at frequency f is approximated by:

$$Loss(f) = Loss_{DC} + (Loss(f_1) - Loss_{DC}) \cdot \sqrt{\frac{f}{f_1}}$$

If in addition the loss at a second frequency f_2 is known (or measured), then the loss can be approximated by:

$$Loss(f) = Loss_{DC} + (Loss(f_1) - Loss_{DC}) \cdot \left(\frac{f}{f_1}\right)^b,\ \text{where}\ b = \frac{\log_{10}\left(\left|\frac{Loss(f_1) - Loss_{DC}}{Loss(f_2) - Loss_{DC}}\right|\right)}{\log_{10}\left(\frac{f_1}{f_2}\right)}$$

In practice, the frequency-dependent part ist often dominant, so that $Loss_{DC}$ can be set to zero. Experimentally, the loss value at DC can be determined in a separate measurement at a very low frequency (f --> 0).

4.6.1.3 Auto Length

The "Auto Length" function adds an electrical length offset to the active trace's receive port, such that the residual delay of the active trace (defined as the negative derivative of the phase response) is minimized across a configurable frequency range. If "Delay" is the selected trace format, the entire trace is shifted in vertical direction and centered on zero. In phase format, the "Auto Length" corrected trace shows the deviation from linear phase.

Length and delay measurement, related settings

"Auto Length" is suited for length and delay measurements on transmission lines.

1. Connect a (non-dispersive) cable to a single analyzer port no. n and measure the reflection factor S_{nn}.
2. In the Channel – [Offset Embed] > "Offset" softtool tab, select "Auto Length".

   The delay is displayed in the "Delay" field, the cable length (depending on the "Velocity Factor") in the "Mech. Length" field.

It is also possible to determine cable lengths using a transmission measurement. Note that "Auto Length" always provides the **single** cable length and the delay for propagation in **one** direction.

The analyzer provides alternative ways for delay measurements:

1. Measure the reflection factor and select Trace – [Format] > "Delay".

   This yields the delay for propagation in forward and reverse direction and should be approx. twice the "Auto Length" result. For transmission measurements, both results should be approx. equal.
2. Measure the reflection factor and select Trace – [Format] > "Phase". Place a marker to the trace and activate Trace – [Trace Config] > "Trace Statistics" > "Phase/El Length".

   This yields the delay in one direction and should be approx. equal to the "Auto Length" result.

The measurement results using trace formats and trace statistics functions depend on the selected delay aperture and evaluation range. "Auto Length" is particularly accurate because it uses all sweep points. For non-dispersive cables, aperture and evaluation range effects are expected to vanish.

Use Trace – [Marker] > "Set by Marker" > Zero Delay at Marker to set the delay at a special trace point to zero.

Preconditions for Auto Length, effect on measured quantities and exceptions

"Auto Length" is enabled if the measured quantity contains the necessary phase information as a function of frequency, and if the interpretation of the results is unambiguous:

- A frequency sweep must be active.
- The measured quantity must be an S-parameter, ratio, wave quantity, a converted impedance or a converted admittance.

The effect of "Auto Length" on S-parameters, wave quantities and ratios is to eliminate a linear phase response as described above. The magnitude of the measured quantity is not affected. Converted admittances or impedances are calculated from the corresponding "Auto Length" corrected S-parameters. Y-parameters, Z-parameters and stability factors are not derived from a single S-parameter, therefore "Auto Length" is disabled.

Auto Length for logical ports

The "Auto Length" function can be used for balanced port configurations as well. If the active test port is a logical port, then the same length offset must be assigned to both physical ports that are combined to form the logical port. If different length offsets have been assigned to the physical ports before, they are both corrected by the same amount.

4.6.1.4 Auto Length and Loss

The "Auto Length and Loss" function determines all offset parameters at the active trace's receive port such that the residual group delay of the active trace (defined as the negative derivative of the phase response) is minimized **and** the measured loss is minimized as far as possible across a configurable frequency range.

"Auto Length and Loss" involves a two-step procedure:

- An "Auto Length" correction modifies the phase of the measured quantity, minimizing the residual group delay. The magnitude of the measured quantity is not affected.
- The automatic loss correction modifies the magnitude of the measured quantity, leaving the (auto length-corrected) phase unchanged.

Preconditions for Auto Length and Loss, effect on measured quantities and exceptions

"Auto Length and Loss" is enabled if the measured quantity contains the necessary phase information as a function of the frequency, and if the interpretation of the results is unambiguous:

- A frequency sweep must be active.
- The measured quantity must be an S-parameter, ratio, wave quantity, a converted impedance or a converted admittance.

The effect of "Auto Length and Loss" on S-parameters, wave quantities and ratios is to eliminate a linear phase response and account for a loss as described above. Converted admittances or impedances are calculated from the corresponding "Auto Length

and Loss" corrected S-parameters. Y-parameters, Z-parameters and stability factors are not derived from a single S-parameter, therefore "Auto Length and Loss" is disabled.

Calculation of loss parameters

The loss is assumed to be given in terms of the DC loss $Loss_{DC}$, the reference frequency f_{ref}, and the loss at the reference frequency $Loss(f_{ref})$. The formula used in the Auto Loss algorithm is similar to the formula for manual entry of the loss parameters (see Section 4.6.1.2, "Definition of loss parameters", on page 198).

The result is calculated according to the following rules:

- The reference frequency f_{ref} is kept at its previously defined value (default: 1 GHz).
- The DC loss c is zero except for wave quantities and for S-parameters and ratios with maximum dB magnitude larger than –0.01 dB.
- "Auto Length and Loss" for a wave quantity centers the corrected dB magnitude as close as possible around 0 dBm.
- "Auto Length and Loss" for S-parameters and ratios centers the corrected dB magnitude as close as possible around 0 dB.

The resulting offset parameters are displayed in the Channel – [Offset Embed] > "Offset" softtool tab.

Auto Length and Loss for balanced ports

The "Auto Length and Loss" function can be used for balanced port configurations as well. If the active test port is a balanced (logical) port, then the same offset parameters must be assigned to both physical ports that are combined to form the logical port. If different offset parameters have been assigned to the physical ports before, they are both corrected by the same amount.

4.6.1.5 Fixture Compensation

"Fixture Compensation" is an automated length offset and loss compensation for test fixtures. The analyzer performs a one-port reflection measurement at each port, assuming the inner contacts of the test fixtures to be terminated with an open or short circuit.

"Fixture Compensation" complements a previous system error correction and replaces a possible manual length offset and loss correction. For maximum accuracy, it is recommended to place the reference plane as close as possible towards the outer test fixture connectors using a full n-port calibration. The "Fixture Compensation" is then carried out in a second step, it only has to compensate for the effect of the test fixture connections.

The following features can further improve the accuracy of the fixture compensation:

- "Direct Compensation" provides a frequency-dependent transmission factor (instead of a global electrical length and loss).

- "Open and Short" causes the analyzer to calculate the correction data from two subsequent sweeps. The results are averaged to compensate for errors due to non-ideal terminations.

Auto Length and Loss vs. Direct Compensation

"Auto Length and Loss" compensation is a descriptive correction type: The effects of the test fixture connection are traced back to quantities that are commonly used to characterize transmission lines.

Use this correction type if your test fixture connections have suitable properties in the considered frequency range:

- The electrical length is approximately constant.
- The loss varies due to the skin effect.

"Direct Compensation" provides a frequency-dependent transmission factor. The phase of the transmission factor is calculated from the square root of the measured reflection factor, assuming a reciprocal test fixture. The sign ambiguity of this calculated transmission factor is resolved by a comparison with the phase obtained in an Auto Length calculation. This compensation type is recommended for test fixture connections that do not have the properties described above.

A "Direct Compensation" resets the offset parameters to zero.

Open/Short vs. Open and Short compensation

A non-ideal "Open" or "Short" termination of the test fixture connections during fixture compensation impairs subsequent measurements, causing an artificial ripple in the measured reflection factor of the DUT. If you observe this effect, an "Open and Short" compensation may improve the accuracy.

"Open and Short" compensation is more time-consuming because it requires two consecutive fixture compensation sweeps for each port, the first with an open, the second with a short circuit. The analyzer automatically calculates suitable averages from both fixture compensation sweeps to compensate for the inaccuracies of the individual "Open and Short" compensations.

Using "Direct Compensation" together with „Open and Short" termination requires close-to-ideal "Open" and "Short" termination with the same length. In most cases, a higher accuracy can be achieved when measuring either "Open" or "Short".

4.6.1.6 Application and effect of offset parameters

Offset and loss parameters can be particularly useful if the reference plane of the calibration cannot be placed directly at the DUT ports, e.g. because the DUT has non-coaxial ports and can only be measured in a test fixture. Offset parameters can also help to avoid a new complete system error correction if a cable with known properties has to be included in the test setup.

- A positive length offset moves the reference plane of the port towards the DUT, which is equivalent to deembedding the DUT by numerically removing a (perfectly matched) transmission line at that port.

- A negative offset moves the reference plane away from the DUT, which is equivalent to embedding the DUT by numerically adding a (perfectly matched) transmission line at that port.

The offset parameters are also suited for length and delay measurements; see Section 4.6.1.3, "Auto Length", on page 199. In contrast to the embedding/deembedding functions (see Section 4.6.2, "Embedding and deembedding", on page 203) the parameters cannot compensate for a possible mismatch in the test setup.

Each offset parameter is assigned to a particular port. The delay parameters affect the phase of all measured quantities related to this port; the loss parameters affect their magnitude. An offset at port 1 affects the S-parameters S_{11}, S_{21}, S_{12}, S_{31}... Some quantities (like the Z-parameters) depend on the whole of all S-parameters, so they are all more or less affected when one S-parameter changes due to the addition of an offset length.

To account for the propagation in both directions, the phase shift of a reflection parameter due to a given length offset is twice the phase shift of a transmission parameter. If, at a frequency of 300 MHz, the electrical length is increased by 250 mm (λ/4), then the phase of S_{21} increases by 90 deg, whereas the phase of S_{11} increases by 180 deg.

Equivalent relations hold for the loss.

If the trace is displayed in "Delay" format, changing the offset parameters simply shifts the whole trace in vertical direction.

The sign of the phase shift is determined as follows:

- A positive offset parameter causes a positive phase shift of the measured parameter and therefore reduces the calculated group delay.
- A negative offset parameter causes a negative phase shift of the measured parameter and therefore increases the calculated group delay.

4.6.1.7 Offset parameters for balanced ports

The offset parameters can be used for balanced port configurations:

- Offset parameters must be assigned to both physical ports of a logical port.
- "Auto Length" corrects the length offset of both physical ports of a logical port by the same amount.

4.6.2 Embedding and deembedding

The R&S ZNB allows you to define virtual networks to be added to/removed from the measurement circuit for a DUT with single-ended or balanced ports. This concept is referred to as embedding/deembedding.

The embedding/deembedding function has the following characteristics:

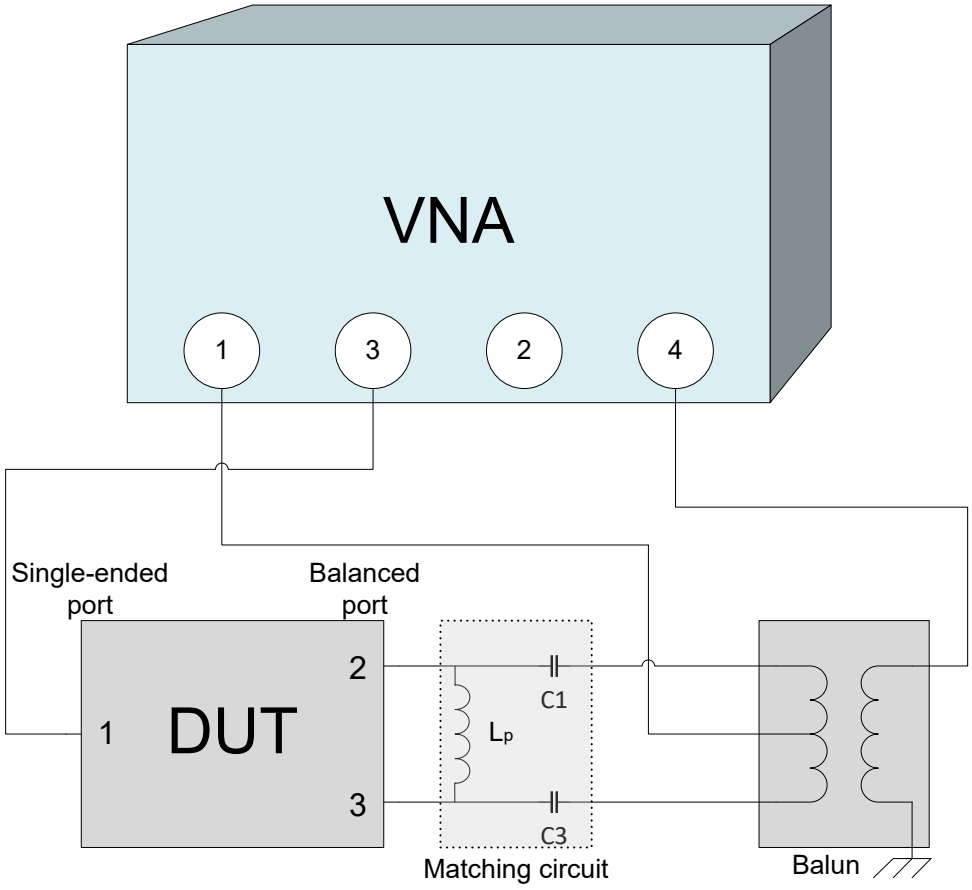
- Embedding and deembedding can be combined with balanced port conversion: the (de-)embedding function is available for single-ended and balanced ports.

- A combination of four-port and two-port networks (not necessarily both) can be applied to balanced ports; two-port networks can be applied to single-ended ports.
- A combination of four-port and two-port networks can be applied to any pair of single-ended ports. Moreover it is possible to combine several port pairs in an arbitrary order (port pair de-/embedding).
- Single-ended and/or balanced port (de-)embedding can be combined with ground loop (de-)embedding. A ground loop models the effect of a non-ideal ground connection of the DUT.
- Transformation networks can be defined by a set of S-parameters stored in a Touchstone file or by an equivalent circuit with lumped elements.
- The same networks are available for embedding and deembedding.

4.6.2.1 Embedding a DUT

To be integrated in application circuits, high-impedance components like Surface Acoustic Wave (SAW) filters are often combined with a matching network. To obtain the characteristics of a component with an added matching network, both must be integrated in the measurement circuit of the network analyzer.

The figure below shows a DUT with a single-ended and a balanced port that is combined with a real matching circuit and a physical unbalance-balance transformer (balun) to be evaluated in a 2-port measurement.



The idea of virtual embedding is to simulate the matching network and avoid using physical circuitry so that the analyzer ports can be directly connected to the input and output ports of the DUT. The matching circuit is taken into account numerically. The analyzer measures the DUT alone but provides the characteristics of the DUT, including the desired matching circuit.

This method provides several advantages:

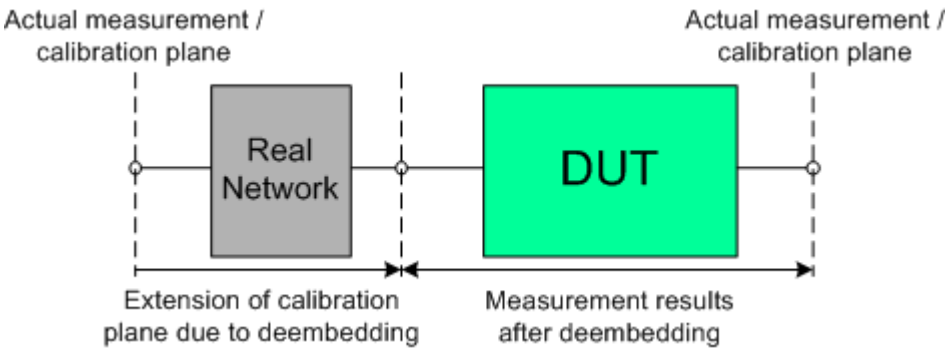
- The measurement uncertainty is not impaired by the tolerances of real test fixtures.
- There is no need to fabricate test fixtures with integrated matching circuits for each type of DUT.
- Calibration can be performed at the DUT's ports. If necessary (e.g. for compensating for the effect of a test fixture), it is possible to shift the calibration plane using length offset parameters.

4.6.2.2 Deembedding a DUT

Deembedding and embedding are inverse operations: A deembedding problem is given if an arbitrary real network connected to the DUT is to be virtually removed to obtain the characteristics of the DUT alone. Deembedding is typically used for DUTs which are not directly accessible because they are inseparably connected to other components, e.g. for MMICs in a package or connectors soldered to an adapter board.

To be numerically removed, the real network must be described by a set of S-parameters or by an equivalent circuit of lumped elements. Deembedding the DUT effectively extends the calibration plane towards the DUT ports, enabling a realistic evaluation of the DUT without the distorting network. Deembedding can be combined with length offset parameters; see Section 4.6.1, "Offset parameters", on page 197.

The simplest case of single port deembedding can be depicted as follows:



4.6.2.3 Circuit models for 2-port networks

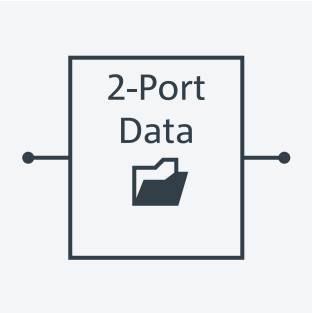
The lumped element 2-port transformation networks for (de-)embedding consist of the following two basic circuit blocks:

- a capacitor connected in parallel with a resistor
- an inductor connected in series with a resistor

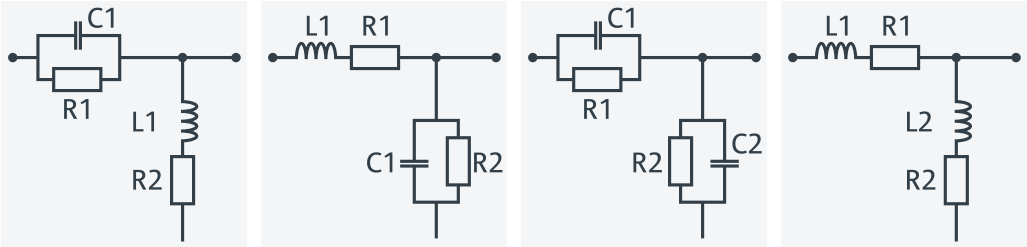
The 2-port transformation networks comprise all possible combinations of 2 basic blocks, where either one block represents a serial and the other a shunt element or

both represent shunt elements. In the default setting the resistors are not effective, since the serial resistances are set to 0 Ω, the shunt resistances are set to 10 MΩ and the shunt inductances are set to 0 Siemens.

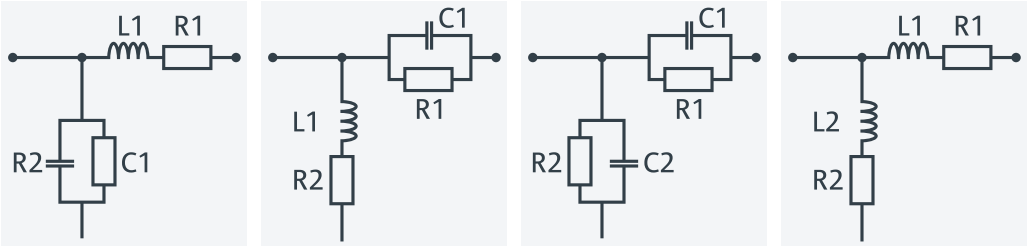
The first network is defined by its S-parameters stored in an imported two-port Touchstone file (`*.s2p`). No additional parameters are required.



The following networks are composed of a serial capacitance C or inductance L (as seen from the test port), followed by a shunt L or C. They are named `Serial C, Shunt L / Serial L, Shunt C / Serial C, Shunt C / Serial L, Shunt L`.

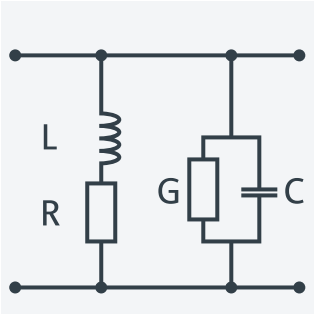


The following networks are composed of a shunt C or L (as seen from the analyzer port), followed by a serial C or L. They are named `Shunt C, Serial L / Shunt L, Serial C / Shunt C, Serial C / Shunt L, Serial L`.



At the GUI, the "capacitance C<i> in parallel with resistance R<i>" circuit blocks can be replaced by equivalent "capacitance C<i> in parallel with conductance G<i>" circuit blocks.

In addition, there is also a `Shunt L, Shunt C` circuit model available, where the shunt C is defined as a capacitance C in parallel with a conductance G:



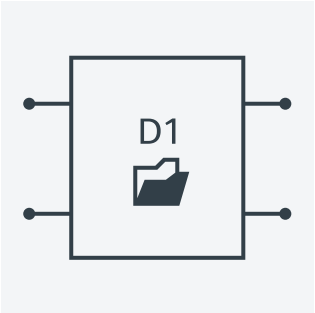
4.6.2.4 Circuit models for 4-port networks

The lumped element 4-port transformation networks for (de-)embedding consist of the following two basic circuit blocks:

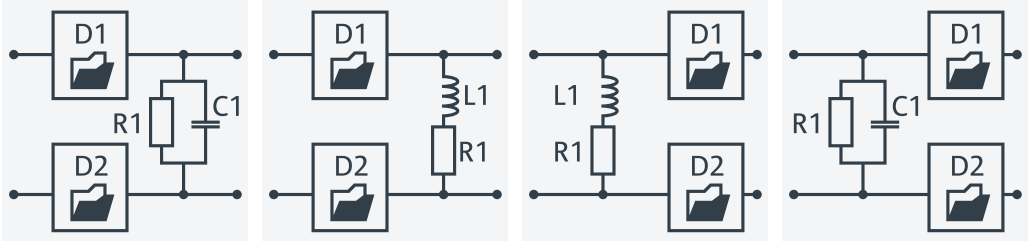
- A capacitor C connected in parallel with a resistor.
- An inductor L connected in series with a resistor.

The transformation networks comprise various combinations of 3 basic circuit blocks, where two blocks represent serial elements, the third a shunt element. In the default setting the resistors are not effective, since the serial Rs are set to 0 Ω, the shunt Rs are set to 10 MΩ. Moreover, the serial elements can be replaced by imported 2-port S-parameters, or the entire transformation network can be described by imported 4-port S-parameters.

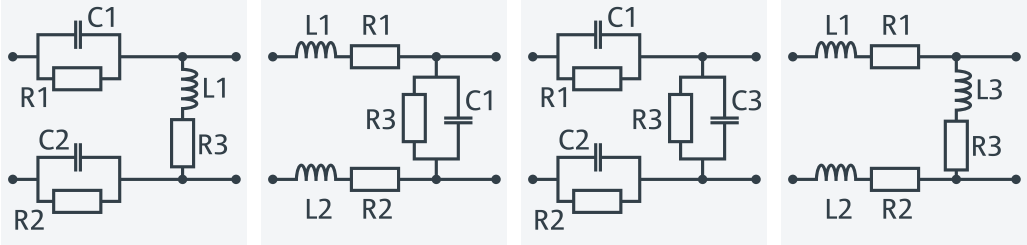
The first network is defined by its S-parameters stored in an imported four-port Touchstone file (`*.s4p`). No additional parameters are required.



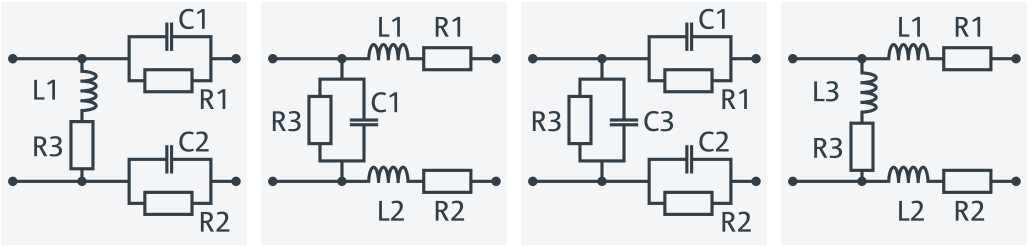
The following networks are composed of a shunt C or L and two serial elements, described by imported 2-port S-parameters. They are named
`Serial 2-port, Shunt C / Serial 2-port, Shunt L / Shunt L, Serial 2-port / Shunt C, Serial 2-port.`



The following networks are composed of two serial Cs or Ls (as seen from the analyzer test port), followed by a shunt C or L. They are named `Serial Cs, Shunt L / Serial Ls, Shunt C / Serial Cs, Shunt C / Serial Ls, Shunt L`.



The following networks are composed of a shunt C or L (as seen from the analyzer test port), followed by two serial Cs or Ls. They are named `Shunt L, Serial Cs / Shunt C, Serial Ls / Shunt C, Serial Cs / Shunt L, Serial Ls`.

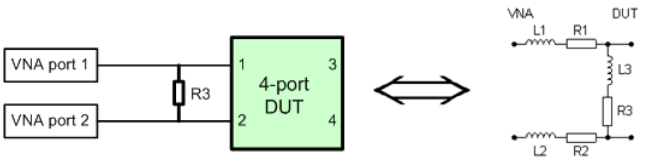


Since FW version 1.93, the "capacitance C<i> in parallel with resistance R<i>" circuit blocks can alternatively be represented as "capacitance C<i> in parallel with conductance G<i>" circuit blocks.

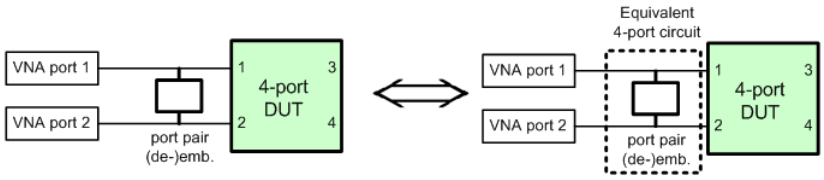
4.6.2.5 Port pair de-/embedding

Port pair de-/embedding extends the functionality of balanced port de-/embedding to pairs of single-ended physical ports. The analyzer uses the 4-port transformation networks known from balanced port de-/embedding, however, each transformation network is assigned to an arbitrary pair of (single-ended) physical ports.

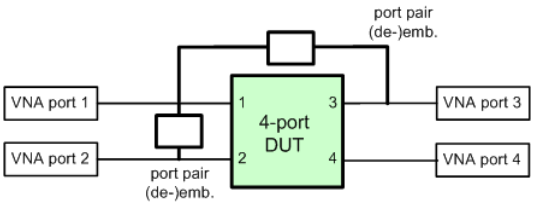
A simple circuit which can be modeled using port pair (de-)embedding is a circuit (e.g. a resistance) between two ports of a DUT. To obtain the circuit in the following figure, select port pair 1,2 and the Serial Ls, Shunt L transformation network with all inductances set to zero (L1 = L2 = L3 = 0 H) and R1 = R2 = 0 Ω.



To model a general (de-)embedding network for ports 1 and 2, select port pair 1, 2 and a 4-Port Touchstone file.



The two port pair (de-)embedding networks in the figure below are based on port pairs 1, 2 and 1, 3 with appropriate sets of 4-port S-parameters.



The R&S ZNB FW handles Port Pair De-/Embedding as a special case of Port set de-/embedding.

4.6.2.6 Port set de-/embedding

The port set de-/embedding feature allows de-/embedding a linear 2m-port network connecting m physical VNA ports to m physical DUT ports (m≥2).

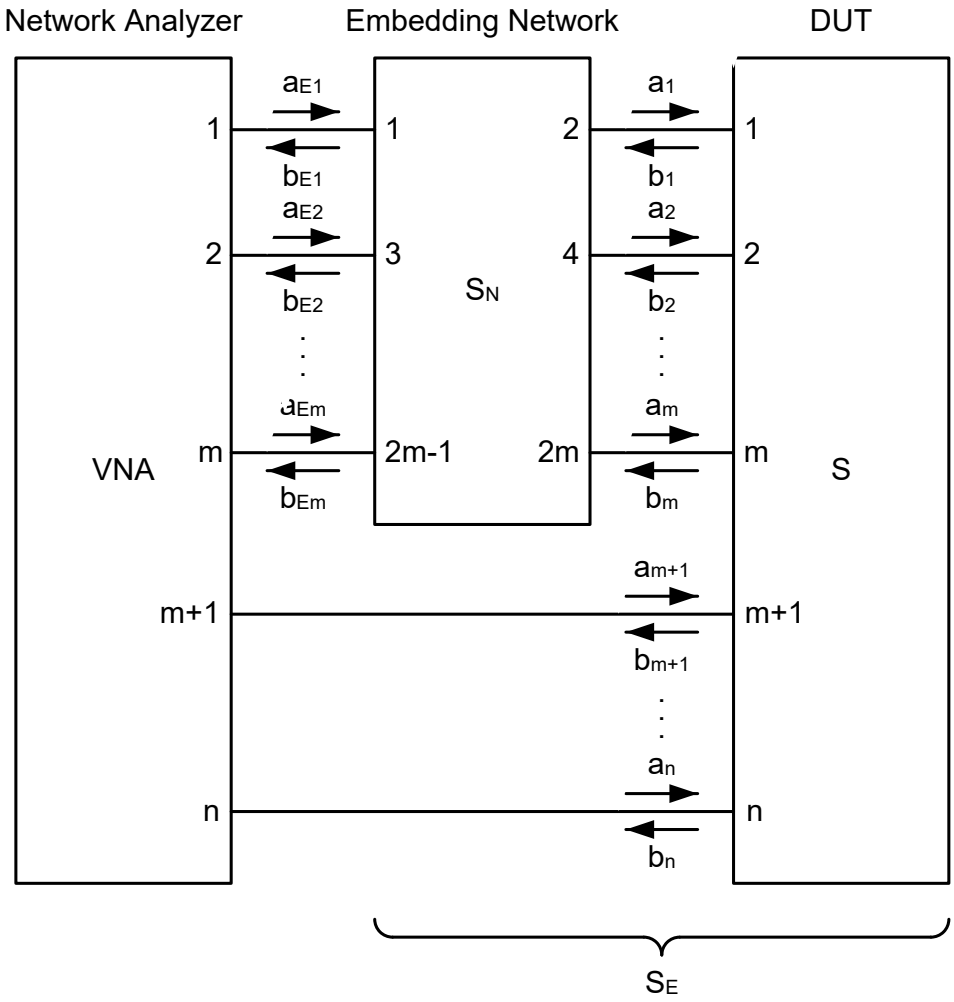


Figure 4-13: Port Set De-/Embedding

As shown in section Combining several de-/embedding networks, port set deembedding is calculated after single-ended deembedding, and the port set embedding step precedes single ended embedding. It is possible to combine a sequence of port sets for deembedding (embedding), each port set having its own transformation network. The effect of port set de-/embedding depends on the port sets themselves but also on their order. The same physical ports can be used repeatedly in different port sets; it is also possible to use the same port set repeatedly.

- For port pairs (i.e. for m=2), the de-/embedding network can be defined either via lumped element model (possibly in combination with s2p Touchstone files) or via a s4p Touchstone file, see Section 4.6.2.5, "Port pair de-/embedding", on page 208. For m>2, there are no predefined lumped element models available; the de-/embedding network has to be defined via a s<2m>p Touchstone file.
- In case the port number conventions of the loaded Touchstone file differ from the numbering scheme depicted in Figure 4-13, it is possible to "Switch Gates". The analyzer interchanges the port numbers when loading the file.

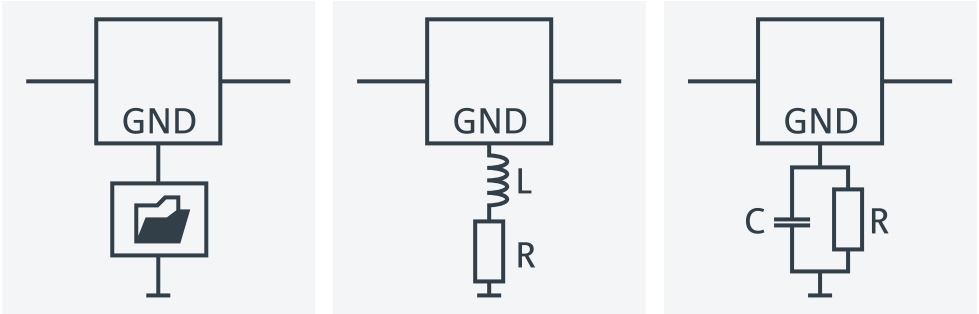


4.6.2.7 Ground loop de-/embedding

A ground loop models the effect of a non-ideal ground connection of the DUT causing a difference in potential between the analyzer's and the DUT's ground reference. A typical and often unavoidable source of ground loops is the parasitic inductance of the ground contacts.

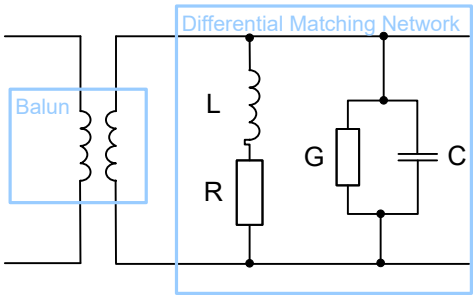
In contrast to the 2-port de-/embedding networks, the ground loop represents a single-wire connection that can be described by a one-port S-matrix. On the other hand, the ground loop de-/embedding algorithm for an n-port DUT involves matrix operations which are based on the complete uncorrected single-ended nxn S-matrix.

The Ground Loop De-/Embedding can be specified via Touchstone s1p file or by parametrizing one of the lumped element models "Shunt L", "Shunt C".



4.6.2.8 Differential match embedding

Differential Match Embedding allows you to simulate the characteristics of balanced DUT ports whose differential mode is balance-unbalance converted and then connected to a port-specific matching circuit.



In contrast to standard balanced embedding (4-port), the matching circuit is only applied to the differential mode port (2-port). It can be specified via a Touchstone s2p file or by parametrizing a lumped "Shunt L, Shunt C" element model.

4.6.2.9 Fixture modeling and deembedding

When performing tasks such as verifying digital high-speed signal structures on printed circuit boards (PCBs), measurements have to be carried out on certain layers without the effects of probes, probe pads, vias, lead-ins and lead-outs. This requires the use of accurate deembedding algorithms to calculate and remove these effects from the measurements, leaving only the result for the area of interest.

With fixture modeling, dedicated third-party tools model the test fixture from measured data using PCB test coupons. So it is not needed to ask the user to define the fixture by parametrizing one of the given lumped circuit models or "somehow" to provide a suitable `snp` file.

The R&S ZNB offers integration of the following third-party tools for fixture modeling:

- AtaiTec's In-situ de-embedding (ISD)
- PacketMicro's Smart fixture de-embedding (SFD)
- Eazy de-embedding based on IEEE 370

More fixture modeling tools may be added in future firmware releases.

Test Setup

The figure shows an example for verifying the high-speed differential signal lines on a PCB.

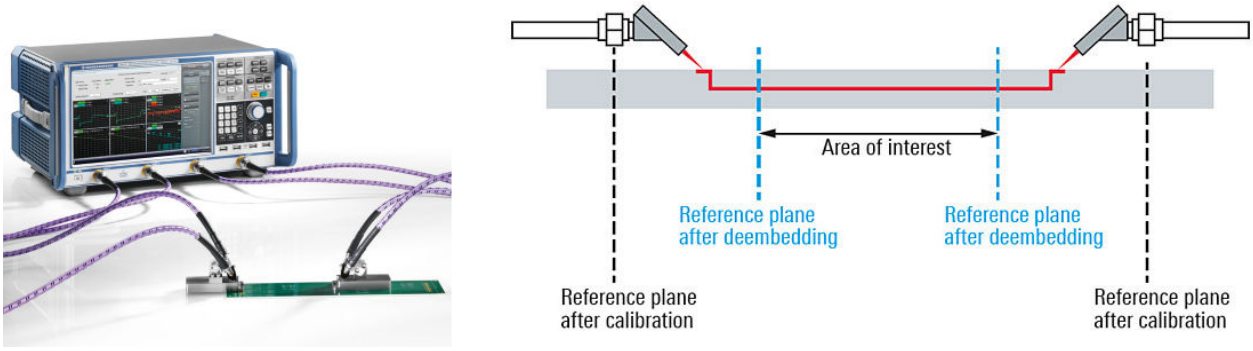


Figure 4-14: Test setup for fixture deembedding

Modeling process

For all supported fixture modeling tools, proceed as follows:

1. Perform a calibration to the fixture connectors.
2. Measure one or more PCB test coupons for the related fixture.
   The results are independent of a particular DUT. Hence they can be reused for subsequent measurements using the same test fixture.
3. Measure the total structure, i.e. the DUT with the fixture.
4. Run the selected third-party tool to calculate the fixture deembedding files.

   After a final confirmation, the calculated deembedding files (`s2p` for single-ended ports, `s4p` for balanced ports) are used for deembedding the DUT at the respective logical port(s).

Test Coupons

The test fixture consists of a set of lead-ins.

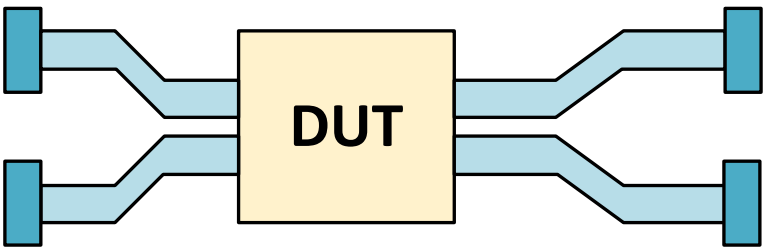


Figure 4-15: DUT with Test Fixture (balanced)

The fixture modeling tools require test coupons for the lead-ins to the DUT and that are either representing a Through, an Open or a Short. If the lead-ins "on the left" and "on the right" side of the DUT ("lead-outs") are not symmetric, different test coupons for the lead-ins and lead-outs are required.

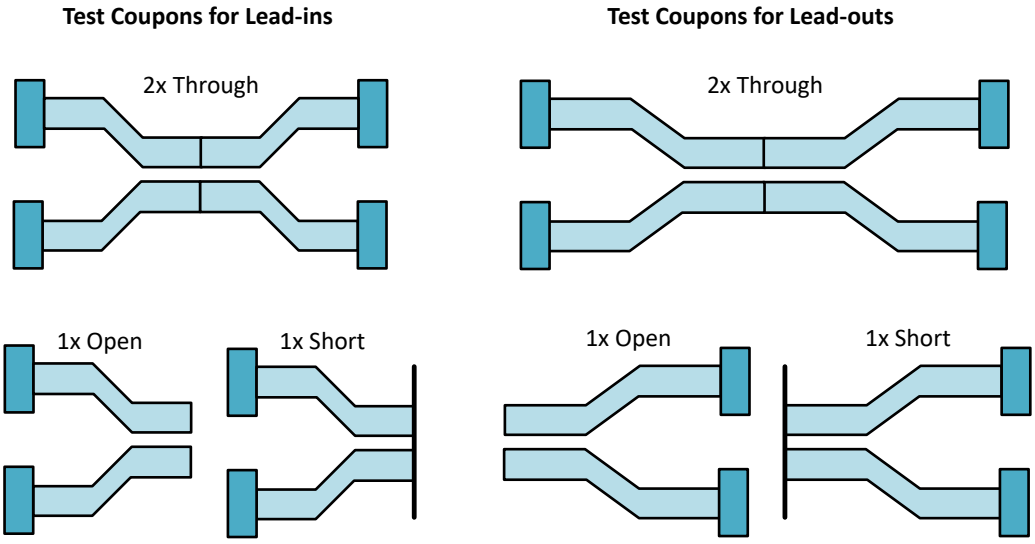


Figure 4-16: Test Coupons (balanced)

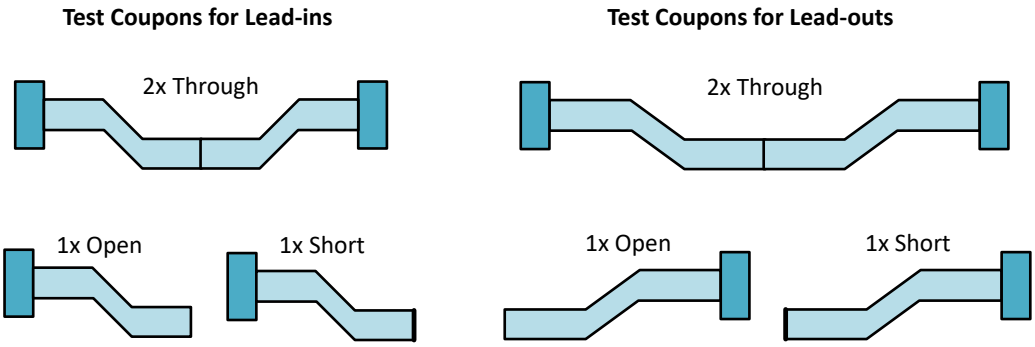


Figure 4-17: Test Coupons (single-ended)

4.6.2.10 Combining several de-/embedding networks

The R&S ZNB allows you to select a combination of networks to be numerically added/ removed at different layers:

- 2-port networks at single-ended physical ports
- 4-port networks at pairs of single-ended physical ports
- 4-port networks at balanced logical ports
- 1-port ground loop networks
- 2-port differential match embedding networks at balanced logical ports

The different steps for deembedding and embedding are carried out in the following order:

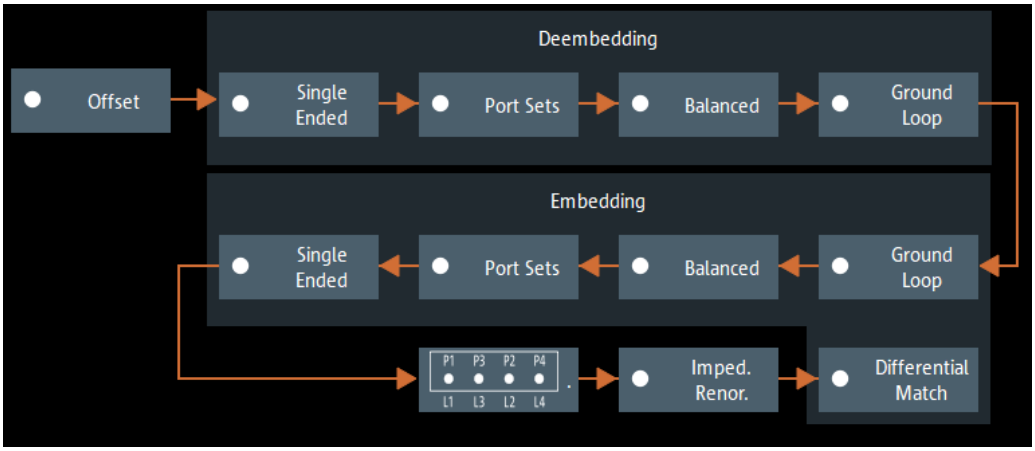


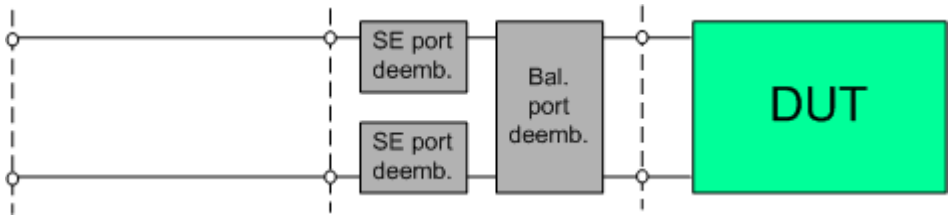
Figure 4-18: De-/Embedding calculation flow

This means that the real networks are removed before virtual networks are added.

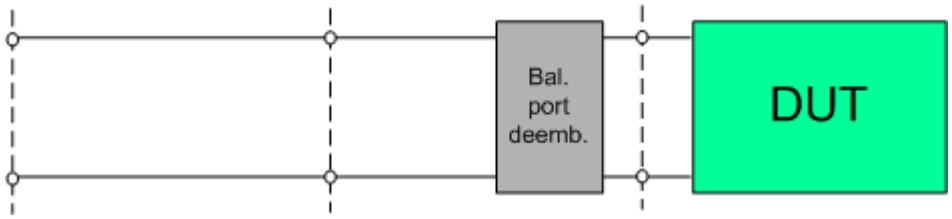
The (de-)embedding steps are carried out in the following order:

1. Single-ended deembedding: every physical port can be deembedded from a single 2-port network
2. Port Set Deembedding: every port set can be deembedded from one or more 4-, 6- or 8-port networks. There is no restriction on the sequence of port pairs and deembedding networks.
3. Balanced Deembedding: every balanced logical port can be deembedded from a single 4-port network
4. Ground Loop Deembedding: the DUT's ground connection can be deembedded from a single 1-port network
5. Ground Loop Embedding: the DUT's ground connection can be embedded in a single 1-port network
6. Balanced Embedding: every balanced logical port can be embedded in a single 4-port network
7. Port Set Embedding: every port set can be embedded in one or more 4-, 6- or 8-port networks. There is no restriction on the sequence of port pairs and embedding networks.
8. Single Ended Embedding: every physical port can be embedded in a single 2-port network
9. Differential Match Embedding: the differential mode of a balanced logical port can be embedded in a single 2-port network

1. Initial situation: DUT embedded in 2-port and 4-port networks (only 1 port shown)



2. Single ended deembedding



3. Balanced port deembedding



4. Balanced port embedding



5. Single-ended port embedding



4.7 Optional extensions and accessories

The instrument can be upgraded with various software and hardware options, providing enhanced flexibility and an extended measurement functionality. The equipped options are listed in the "Options" tab of the Info dialog (System – [Setup] > "Setup" > "Info").

Table 4-18: Software options (K-options)

Option	Option name
R&S ZNB-K2	Time domain analysis
R&S ZNB-K3	Distance-to-fault measurements
R&S ZNB-K4	Frequency conversion measurements
R&S ZNB-K14	Intermodulation measurements
R&S ZNB-K17	Receiver bandwidth 10 MHz
R&S ZNB-K19	Frequency resolution 1 mHz
R&S ZNB-K20	Extended time domain analysis
R&S ZNB-K30	Noise figure measurement
R&S ZNB-K50, R&S ZNB-K50P	Measurement uncertainty analysis
R&S ZNB-K100	SNP assistant
R&S ZNB-K210	Eazy de-embedding based on IEEE 370
R&S ZNB-K220	In-situ de-embedding
R&S ZNB-K230	Smart fixture de-embedding
R&S ZNB-K231	Delta-L 4.0 PCB characterization
R&S ZNB-K980	Health and utilization monitoring service (HUMS)

Permanent vs. portable licenses

All software options are available as permanent and portable license. A permanent license (variant 02) is bound to a particular instrument. A portable license (variant 07) can be moved between different instruments of the applicable type.

To move a software license, visit https://www.rohde-schwarz.com/license-manager, run the "Deactivate or move licenses" application, and follow its on-screen instructions.

Table 4-19: Hardware options (B-options)

Option	Option name
R&S ZNB-B1	Bias tees
R&S ZNBxx-B2	Second internal generator
R&S ZNB-B4	Precision frequency reference
R&S ZNB-B10	GPIB interface
R&S ZNB-B12	Device Control
R&S ZN-B14	Handler I/O (universal interface)
R&S ZN-B15	Internal RFFE GPIO interface (1st generation R&S ZNB only)
R&S ZN-Z15	External RFFE GPIO interface
R&S ZNB-B19	Additional removable system drive

Option	Option name
R&S ZNBxx-B22, -B24	Extended power range
R&S ZNB4|8-B31, -B32, -B33, -B34	Receiver step attenuators
R&S ZVAB-B44	USB-to-IEC/IEEE adapter
R&S ZNBxx-B52, -B54	Extended dynamic range
R&S ZNB40-B73, -U73	R&S ZNB40 reflector module with ESD limiters
R&S ZNB-B81	DC inputs

For a complete list of options, accessories, and extras refer to the product brochure or to the "Options" section of the R&S ZNB product pages on the Internet.

The following sections provide an introduction to the software and hardware options described in this documentation. The use of external power meters, generators and switch matrices does not require any additional hardware or software options; it is described at the end of the chapter.

4.7.1 Time domain analysis

Option R&S ZNB-K2

The network analyzer measures and displays complex S-parameters and other quantities as a function of the frequency. The measurement results can be filtered and mathematically transformed to obtain the time domain representation, which often gives a clearer insight into the characteristics of the DUT.

Time domain transforms can be calculated in band pass or low pass mode. For the latter, the analyzer offers the impulse and step response as two alternative transformation types. A wide selection of windows can be used to optimize the time domain response and suppress side lobes due to the finite sweep range. Moreover, it is possible to eliminate unwanted responses using a time gate and transform the gated result back into the frequency domain.

For a detailed discussion of the time domain transformation including many examples, refer to the application note 1EP83 which is available on the R&S internet at http://www.rohde-schwarz.com/appnotes/1EP83.

Trace selection

Select a reflection parameter S_{ii} to perform a time domain reflectometry (TDR) measurement, e.g. to detect cable faults. To measure the impedance of a network, you can select a converted impedance parameter $Z \leftarrow S_{ii}$.

Select a transmission parameter S_{ij} ($i \neq j$) to perform a time domain transmission (TDT) measurement, e.g. for measuring antennas.

- For TDR traces and if distance is used as the x axis, the x value indicates the **one-way** path to the reflection.
- To export time domain traces, you have to use formatted values (ASCII, Matlab). See Section 4.4.2.3, "Finding the best file format", on page 152.

4.7.1.1 Chirp z-transformation

The Chirp z-transformation that the analyzer uses to compute the time domain response is an extension of the (inverse) Fast Fourier Transform (FFT). Compared to the FFT, the number of sweep points is arbitrary (not necessarily an integer power of 2), but the computation time is increased by approx. a factor of 2. This increased computation time is usually negligible compared to the sweep times of the analyzer.

The following properties of the Chirp z-transformation are relevant for the analyzer settings:

- The frequency points must be equidistant.
- The time domain response is repeated after a time interval which is equal to $\Delta t = 1/\Delta f$, where Δf is the spacing between two consecutive sweep points in the frequency domain. For a sweep span of 4 GHz and 201 equidistant sweep points, $\Delta f = 4\ \text{GHz}/200 = 2 * 10^7$ Hz, so that $\Delta t = 50$ ns. Δt is termed measurement range (in time domain) or unambiguous range.

Additional constraints apply if the selected Chirp z-transformation is a low-pass transformation.

4.7.1.2 Band pass and low pass mode

The analyzer provides two different types of time domain transforms:

- Band pass mode: The time domain transform is based on the measurement results obtained in the sweep range between any set of positive start and stop values. The sweep points must be equidistant. No assumption is made about the measurement point at zero frequency (DC value). The time domain result is complex, with an undetermined phase depending on the delay of the signal.
- Low pass mode: The measurement results are continued towards f = 0 (DC value) and mirrored at the frequency origin so that the effective sweep range (and thus the response resolution) is more than doubled. Together with the DC value, the condition of equidistant sweep points implies that the frequency grid must be harmonic. Due to the symmetry of the trace in the frequency domain, the time domain result is harmonic.
  See also Section 4.7.1.4, "Harmonic grid", on page 221.
  Two different types of response are available in low pass mode; see below.

Table 4-20: Comparison of band pass and low pass modes

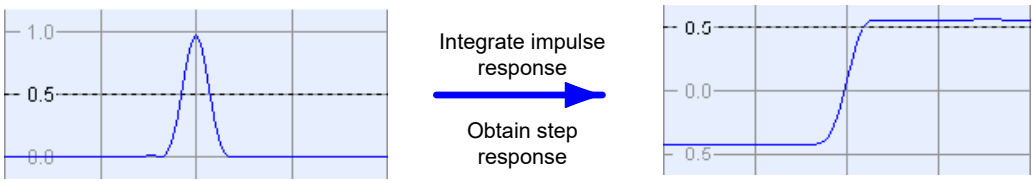
Transform type	Band pass	Low pass
Advantages	Easiest to use: works with any set of equi-distant sweep points	Higher response resolution (doubled) Includes information about DC value Real result Impulse and step response
Restrictions	No step response Undetermined phase	Needs harmonic grid
Use for...	Scalar measurements where the phase is not needed DUTs that do not operate down to f = 0 (e.g. pass band or high pass filters)	Scalar measurements where the sign is of interest DUTs with known DC value

Impulse and step response

In low pass mode, the analyzer can calculate two different types of responses:

- The impulse response corresponds to the response of a DUT that is stimulated with a short pulse.
- The step response corresponds to the response of a DUT that is stimulated with a voltage waveform that transitions from zero to unity.

The two alternative responses are mathematically equivalent; the step response can be obtained by integrating the impulse response:



The step response is recommended for impedance measurements and for the analysis of discontinuities (especially inductive and capacitive discontinuities). The impulse response has an unambiguous magnitude and is therefore recommended for most other applications.

4.7.1.3 Windows in the frequency domain

The finite sweep range in a frequency domain measurement with the discontinuous transitions at the start and stop frequency broadens the impulses and causes side lobes (ringing) in the time domain response. The windows offered in the "Define Transform" dialog can reduce this effect and optimize the time domain response. The windows have the following characteristics:

Table 4-21: Properties of standard frequency windows

Window	Side lobe suppression	Relative impulse width	Best for...
No Profiling (Rectangle)	13 dB	1	–
Low First Side lobe (Hamming)	43 dB	1.4	Response resolution: separation of closely spaced responses with comparable amplitude
Normal Profile (Hann)	32 dB	1.6	Good compromise between pulse width and side lobe suppression
Steep Falloff (Bohman)	46 dB	1.9	Dynamic range: separation of distant responses with different amplitude
Arbitrary Side lobes (Dolph-Chebyshev)	User defined between 10 dB and 120 dB	1.2 (at 32 dB side lobe suppression)	Adjustment to individual needs; tradeoff between side lobe suppression and impulse width

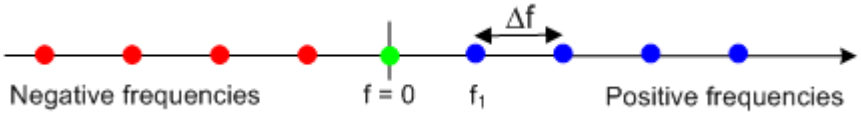
Table 4-22: Additional frequency windows

Window	Description
Blackman	The Blackman window is a fixed window function that offers very low sidelobe levels, providing approximately 96 dB of sidelobe reduction. In the time-domain response, this window significantly reduces ripples.
Kaiser	The Kaiser window is an adjustable window function. The smoothness factor, ranging from 0 to 1, corresponds to the β value in the original Kaiser window. A higher smoothness factor results in more aggressive sidelobe reduction.
Gaussian	The Gaussian window is an adjustable window function. The smoothness factor, ranging from 0 to 1, inversely corresponds to the standard deviation (σ) value in the original Gaussian function.
Exponential	The Exponential window is an adjustable window function. The smoothness factor, ranging from [0, 1], directly corresponds to the decay rate (τ).
Causal	Causal filter with definable rise time

4.7.1.4 Harmonic grid

A harmonic grid is formed by a set of equidistant frequency points f_i (i = 1...n) with spacing Δf and the additional condition that f_1 = Δf. In other words, all frequencies f_i are set to harmonics of the start frequency f_1.

If a harmonic grid – including the DC value (f = 0) – is mirrored to the negative frequency range, the result is again an equidistant grid.

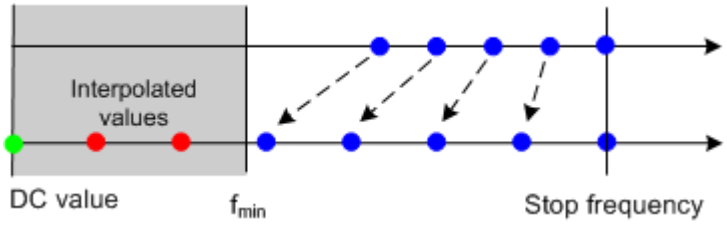


The point symmetry about the DC value makes harmonic grids suitable for low-pass time domain transformations.

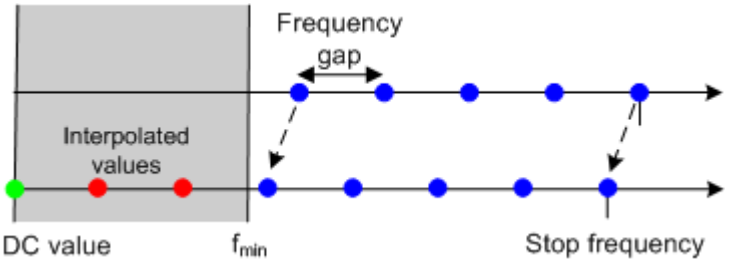
Visualization of the harmonic grid algorithms

The R&S ZNB provides three different algorithms for harmonic grid calculation. The three harmonic grids have the following characteristics:

- Keep "Stop Frequency and Number of Points" means that the stop frequency and the number of sweep points is maintained. The sweep points are redistributed across the range between the minimum frequency of the analyzer and the stop frequency; the step width can be increased.

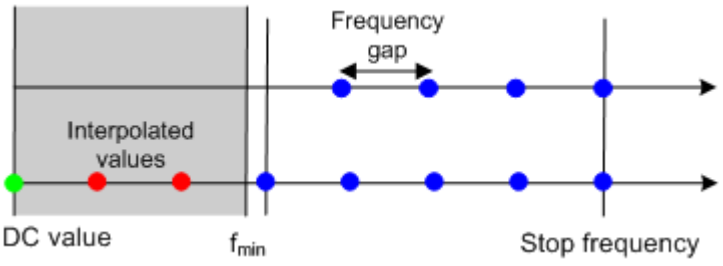


- Keep "Frequency Gap and Number of Points" means that the number of sweep points and their relative spacing is maintained. If the start frequency of the sweep is sufficiently above the f_{min}, the entire set of sweep points is shifted towards lower frequencies so that the stop frequency is decreased.



  If the start frequency of the sweep is close to f_{min}, then the sweep points can have to be shifted towards higher frequencies. If the last sweep point of the calculated harmonic grid exceeds the maximum frequency of the analyzer, then an error message is displayed, and another harmonic grid algorithm must be used.

- Keep "Stop Frequency and Approximate Frequency Gap" means that the stop frequency is maintained and the number of sweep points is increased until the range between f_{min} and the stop frequency is filled. The frequency gap is approximately maintained.



The figures above are schematic and do not comply with the conditions placed on the number of sweep points and interpolated/extrapolated values.

The harmonic grids cannot be calculated for any set of sweep points. If the minimum number of sweep points is smaller than 5, then the interpolation/extrapolation algorithm for additional sweep points does not work. The same is true if the number of sweep points or stop frequency exceeds the upper limit. Besides, the ratio between the sweep range and the interpolation range between f = 0 and f = f_{min} must be large enough to ensure accurate results. If the sweep range for the harmonic grid exceeds the frequency range of the current system error correction, a warning is displayed.

Finding the appropriate algorithm

The three types of harmonic grids have different advantages and drawbacks. Note that for a bandpass transformation the grid parameters have the following effect:

- A wider sweep range (i.e. a larger bandwidth) increases the time domain resolution.
- A smaller frequency gap extends the unambiguous range.
- A larger number of points increases the sweep time.

With default analyzer settings, the differences between the grid types are small. The following table helps you find the appropriate grid.

Table 4-23: Properties of grid types

Grid type: Keep	Sweep time	Time domain resolution	Unambiguous range	Algorithm fails if...
Stop freq. and no. of points	→	↑	↓	–
Freq. gap and no. of points	→	→	→	Stop frequency beyond upper frequency limit
Stop freq. and approx. freq. gap	↑	↑	→	Number of sweep points beyond limit

4.7.1.5 Time gates

A time gate is used to eliminate unwanted responses that appear on the time domain transform. An active time gate acts on the trace in time domain and in frequency domain representation.

The properties of the time gates are analogous to the properties of the frequency domain windows. The following table gives an overview:

Table 4-24: Properties of time gates

Window	Side lobe suppression	Passband ripple	Best for...
Steepest Edges (Rectangle)	13 dB	0.547 dB	Eliminate small distortions near the useful signal, if demands on amplitude accuracy are low
Steep Edges (Hamming)	43 dB	0.019 dB	Good compromise between edge steepness and side lobe suppression

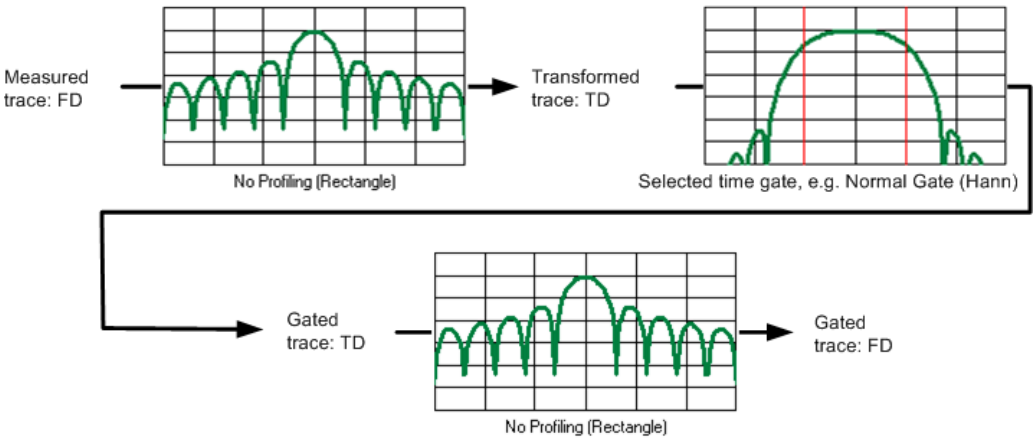
Window	Side lobe suppression	Passband ripple	Best for...
Normal Gate (Hann)	32 dB	0.032 dB	Good compromise between edge steepness and side lobe suppression
Maximum Flatness (Bohman)	46 dB	0 dB	Maximum attenuation of responses outside the gate span
Arbitrary Gate Shape (Dolph-Chebyshev)	User defined between 10 dB and 120 dB	0.071 dB	Adjustment to individual needs; tradeoff between side lobe suppression and edge steepness

Time-gated frequency domain trace

The trace in the frequency domain depends on the state of the "Time Gate":

- If the gate is disabled, the frequency domain (FD) trace corresponds to the measured sweep results before the time-domain transformation.
- If the gate is enabled, the displayed frequency domain trace is calculated from the time domain (TD) trace which is gated and transformed back into the frequency domain.

The analyzer uses fixed "No Profiling (Rectangle)" window settings to transform the measured trace into time domain. The TD trace is gated using the selected time gate. The gated trace is transformed back into frequency domain using a "No Profiling (Rectangle)" window.



The shape, width and position of the time gate affect the gated frequency domain trace. The window type selection in the "Define Transform" dialog is ignored. The selected window is used again when the TD trace is displayed ("Time Domain: On").

The rectangular "No Profiling (Rectangle)" windows minimize numerical inaccuracies near the boundaries of the measured frequency span. In the limit where the effect of the time gate vanishes (e.g. a gate of type "Notch" with small width), the time gated trace is equal to the original measured trace.

4.7.1.6 Time domain S_{VSWR} measurements

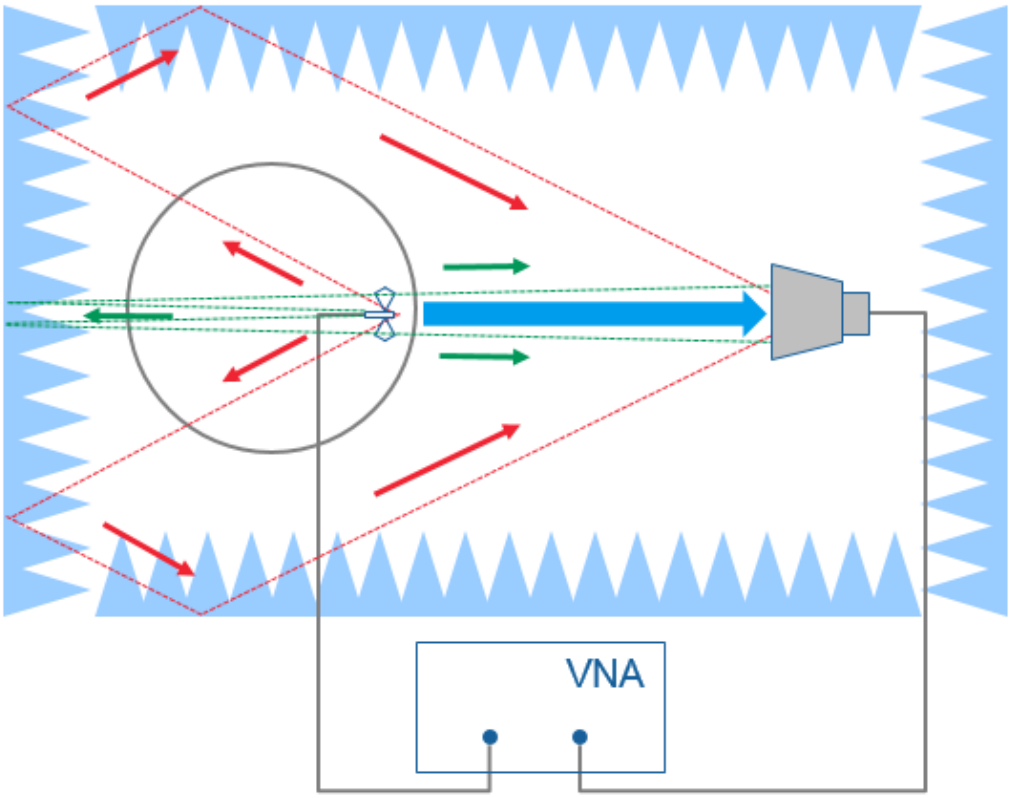
The vector network analyzer R&S ZNB with time domain option K2 supports TD site VSWR measurements.

For the frequency range 1 GHz to 18 GHz, this method of S_{VSWR} measurement is proposed in standard ANSI C63.25.

Hence an instrument with upper frequency limit above 18 GHz that is equipped with time domain option K2 supports TD site VSWR measurements in accordance with ANSI C63.25.

EMC Test Site Validation

EMC test sites for radiated emission measurements rely on free-space conditions to minimize the influence of reflections on the received signal. Practically, near free-space conditions are achieved by shielded enclosures fully lined with RF absorbing material. *Site validation* determines deviations from free-space conditions that must meet the acceptance criterion for making EMC compliance measurements in a FAR (= fully anechoic room). It is performed by measuring the *site voltage standing-wave ratio* (S_{VSWR}), which is the ratio of maximum received signal to minimum received signal, caused by interference between direct (intended) and reflected signals.



blue arrow = direct signal
red and green arrows = reflected signal

TD S_{VSWR} Calculation

The ANSI time-domain method relies on a complex transmission measurement (S_{21}) using a vector network analyzer (VNA).

A time-domain transformation of the frequency domain data shows the impulse response between the two antennas. Since the direct antenna-to-antenna response is related to the shortest distance, the earliest impulse is the direct antenna response. Reflections from the test site are from farther distance away, thus come at later time. It is therefore possible to separate the direct antenna response $S_{21, direct}$ from reflections $S_{21, reflected}$ by using time gating and to calculate

TD S_{VSWR} = (1+r) / (1-r) ,

where r = $S_{21,reflected}$ / $S_{21,direct}$.

Measurement Procedure for ANSI C63.25

See the Rohde & Schwarz Application Card "Fast validation of EMC Test Sites above 1 GHz with Time-domain S_{VSWR}", published on the Rohde & Schwarz internet site.

TD S_{VSWR} measurements in accordance with ANSI C63.25 require frequencies of 20 GHz or higher.

4.7.1.7 Extended time domain analysis

Option R&S ZNB-K20

Option K20 extends the basic Time Domain representation capabilities of option K2 by signal integrity testing functionality in the time domain.

Simulated eye diagram

With the impulse response calculated from the measured S-parameters using the inverse Fourier transform, it is possible to predict the system response to arbitrary time domain signals by calculating the convolution of the input signal with the impulse response.

With option R&S ZNB-K20, the R&S ZNB firmware implements a "virtual" signal generator that is able to generate multilevel PAM signals (NRZ, PAM-4, PAM-8, PAM-16), including a simulated low pass behavior. The simulated eye diagram using the DUT's measured S parameters gives an extensive overview regarding the signal integrity of the system.

Further building blocks for pre-emphasis, jitter, noise and equalization allow you to synthesize effects of transmitter and receiver parts in the transmission system.

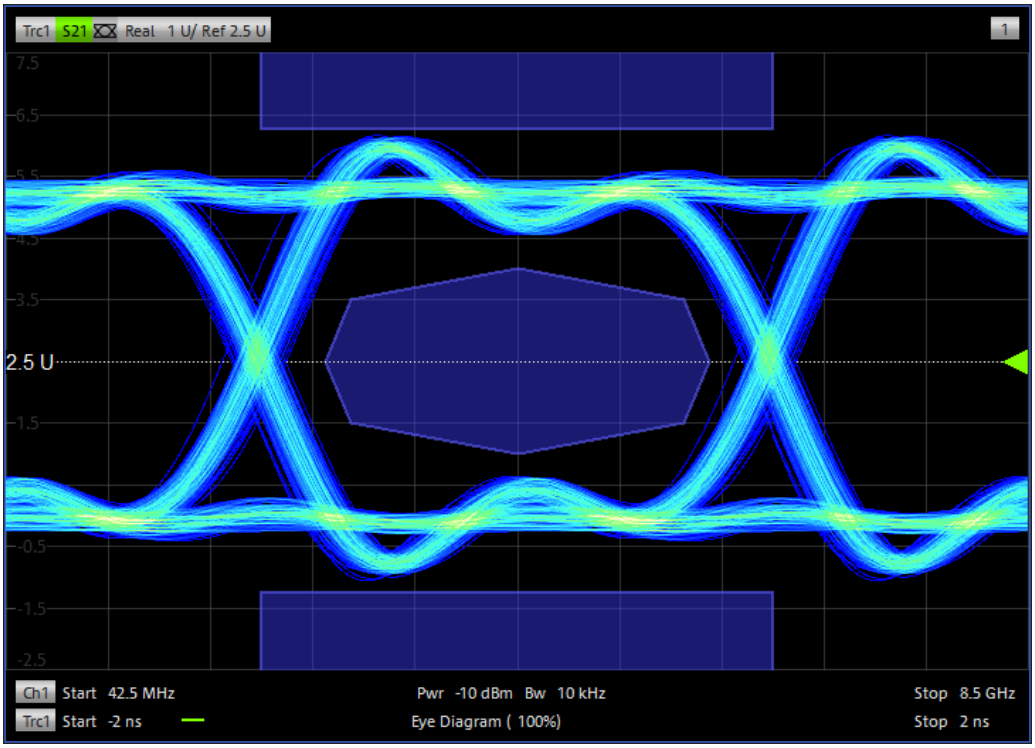


Figure 4-19: Eye diagram (NRZ modulated)

The analyzer proceeds as follows:

1. Perform a frequency sweep.
2. Calculate the impulse response based on the results of the preceding frequency sweep.
3. Based on the impulse response calculated in step 2:
   a) Simulate the eye diagram
   b) Calculate the Eye diagram results
   c) Evaluate the Eye mask test (if enabled).
4. In continuous sweep mode, go back to step 2

- The simulation progress is shown in the channel list below the eye diagram
- In continuous sweep mode, the analyzer keeps on sweeping while the eye diagram is being simulated and evaluated

Eye diagram results

The properties of the simulated eye diagram can be displayed as an info field or exported to an ASCII file.

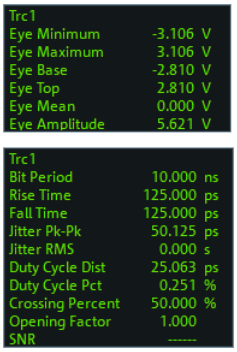


Figure 4-20: Eye diagram result info field

The following results are available:

- **Eye Minimum** and **Eye Maximum**
  These values represent the minimum and maximum outputs of the eye diagram processing. These values include any over- and under-shoots seen during the symbol transitions.
- **Eye Base**, **Eye Top**, **Eye Mean**, and **Eye Amplitude**
  "Eye Base" and "Eye Top" represent the averaged voltages observed for the low and high voltage bits, respectively. "Eye Mean" is the average of "Eye Top" and "Eye Base", "Eye Amplitude" is the difference between "Eye Top" and "Eye Base".
- **Eye Height**
  This result indicates the effects of noise in reducing the vertical eye opening. It is defined as ("Eye Top" - $3\sigma_{Top}$) - ("Eye Base" + $3\sigma_{Base}$), where σ_{Top} and σ_{Base} denote the standard deviations of the observed high and low voltage levels, respectively.
- **Eye Width**
  This result indicates the effects of jitter in reducing the horizontal eye opening. It is defined as "Bit Period" - 2 · 3 · "Jitter RMS".
- **Bit Period**
  The inverse of the data rate.
- **Rise Time** and **Fall Time**
  The time it takes the rising (falling) edge of the eye to go from x% (y%) of the "Eye Amplitude" to y% (x%) of the "Eye Amplitude" ($0 \le x \le y \le 100$, typically x=10 and y=90).
- **Jitter Pk-Pk/RMS**
  These values measure the excursion of the 50% point of the rising edge in the horizontal (time) direction: "Jitter Pk-Pk" is the distance between the peak values; "Jitter RMS" is the RMS value of the excursions.
- **Duty Cycle Dist/Pct**
  These values measure the time separation between the rising and falling edge at the 50% level of the eye diagram. "Duty Cycle Dist" is the absolute distance, "Duty Cycle Pct" gives the same value as a percentage of the "Bit Period".
- **Crossing Percent**
  The Crossing Height is the height above "Eye Base" where the rising and falling edges cross, "Crossing Percent" gives the same value as a percentage of the "Eye Amplitude".

- **Opening Factor**
  This result is a measure of the effects of amplitude noise on the vertical eye opening. Accordingly, it is equal to
  (("Eye Top" - σ_{Top}) - ("Eye Base" + σ_{Base})) / "Eye Amplitude"
- **SNR**
  Relates the "Eye Amplitude" to the noise level. Accordingly, it is equal to
  "Eye Amplitude" / (σ_{Top} + σ_{Base})

Eye diagram results are only available for NRZ modulated signals.

Eye mask test

Furthermore, the eye diagram simulation allows testing against user-defined eye masks:

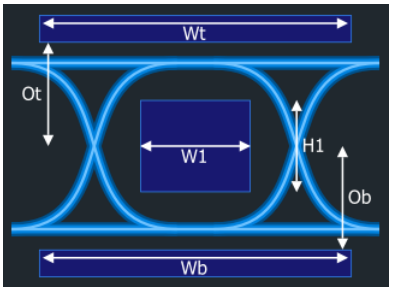


Figure 4-21: User-defined Eye Mask

Eye mask tests are only available for NRZ modulated signals.

Rise time measurement

From the measured S parameters, the step responses can also be calculated using the inverse Fourier transform. The rise time is the time that the step response takes to rise from x% to y% (0≤x≤y≤100) of the resulting step size, typically from 10% to 90%.

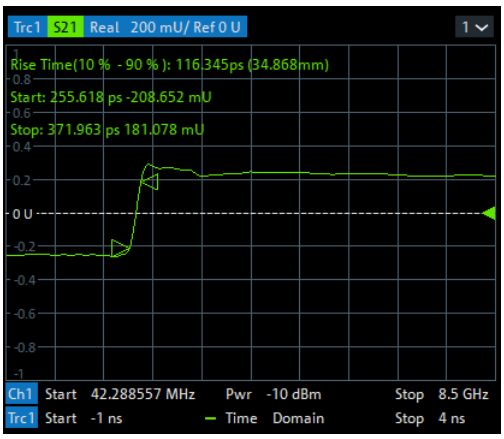


Figure 4-22: Rise time measurement

Skew measurement

The skew measurement allows you to compare the step responses calculated from different traces measured on the same analyzer channel. The skew is calculated as the "delta time" (or length) at user-defined percentage of the step size.

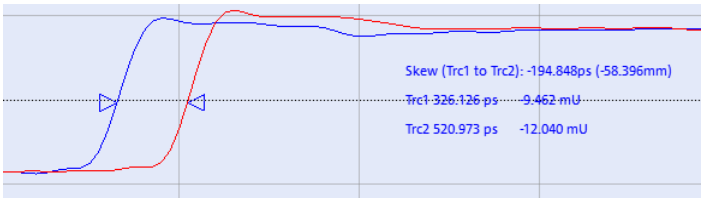


Figure 4-23: Skew measurement

The measured skew can be used to evaluate the quality of a balanced transmission line ("intra-pair skew").

Limit test

Testing against a user-defined skew limit is also supported.

Pulse response measurement

The pulse response measurement calculates the pulse response of a simulated signal from the measured S-parameters. Key characteristics such as pre-cursors, main cursor, post-cursors and intersymbol interference (ISI) are derived.



DUT-centric setup

As far as possible the required settings are related to time domain properties of the DUT, which is more intuitive for users of digital oscilloscopes that are not familiar with vector network analyzers.

The Automatic Harmonic Grid from the option R&S ZNB-K2 "Time domain Analysis" is used to ease setting up the frequency grid for all measurements provided by the option R&S ZNB-K20.

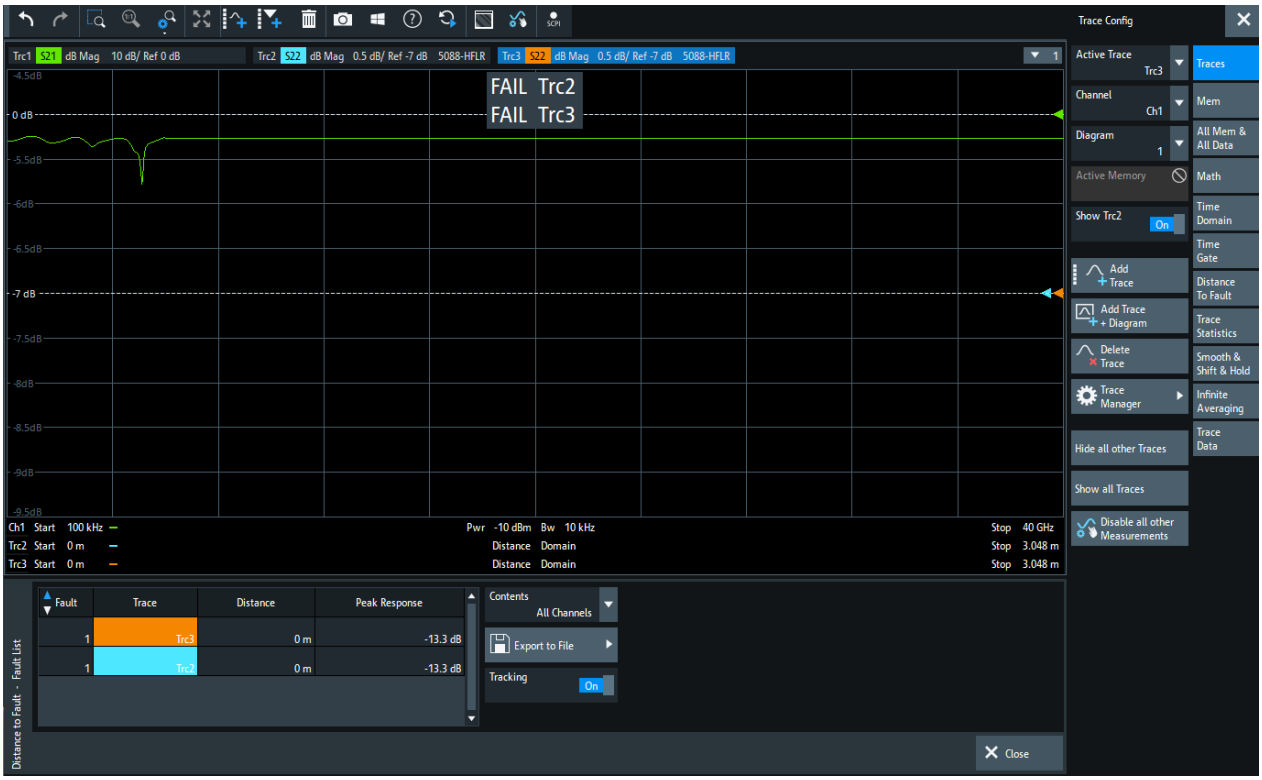
4.7.2 Distance-to-fault measurements

Option R&S ZNB-K3

With option R&S ZNB-K3 "Distance-to-Fault", the R&S ZNB can locate faults and discontinuities on cables and transmission lines.

Faults produce peaks on the impulse response (in the time domain) that is calculated from the measured reflection S-parameter trace via inverse Fourier transformation. The distance between the reference plane and the fault can then be calculated from the propagation time, accounting for the electrical properties of the transmission line. Moreover it is possible to define the peaks to be considered as being caused by a fault.

The following example shows the reflection at the end of a faultless open cable connected to test port 2. A full one-port calibration ensures that the distance is measured from the test port position (reference plane = distance zero). The horizontal position of marker "M2" corresponds to the cable length of 1417 mm, the additional peak at position "M1" is due to a bad connection between the cable and the test port.



Adjust the frequency sweep to the length of the transmission line and the expected distance to fault:

- The maximum distance that can be measured is proportional to the number of sweep points. The larger the number of sweep points, the longer the maximum distance between the calibrated reference plane and the fault to be located.
- The maximum distance that can be measured is inversely proportional to the frequency span. The smaller the frequency span, the longer the maximum distance between the calibrated reference plane and the fault to be located.
  However, a smaller frequency span comes at the cost of a coarser distance resolution.

The R&S ZNB can automatically determine a suitable number of sweep points for a given maximum distance to fault ("Auto Number of Points"). Moreover it is possible to define which of the peaks are considered as being due to a fault, and to draw up and export a list of the detected faults.

A more flexible approach with additional configuration possibilities is available with time domain option R&S ZNB-K2 (see Section 4.7.1, "Time domain analysis", on page 218).

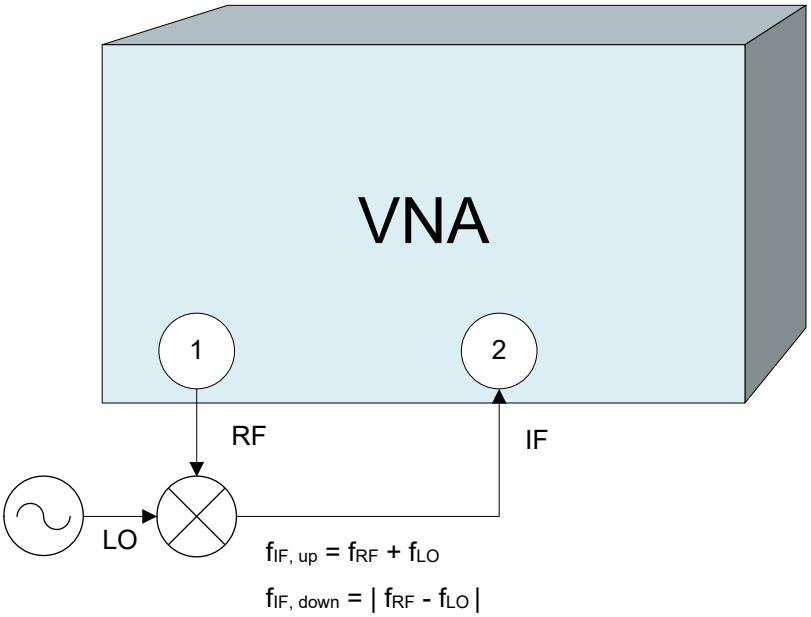
4.7.3 Frequency conversion measurements

R&S ZNB-K4

With option R&S ZNB-K4, the port frequencies can be configured independently. Those arbitrary port settings represent a major extension to the analyzer's measurement functionality; in particular they allow you to measure frequency-converting DUTs.

The source and receive frequencies of the ports are always equal.

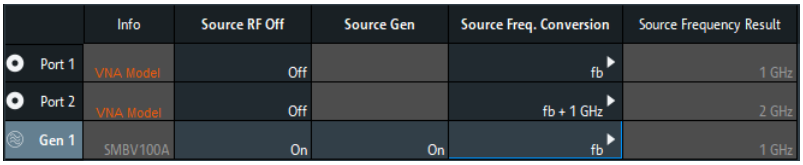
The following figure shows a transmission measurement on a mixer.



In this example, the transmission parameter S_{21} corresponds to the mixer's conversion gain.

For an I/Q mixer, its two IF ports ("I" and "Q") are connected to two VNA ports, with the same frequency conversion.

Before measuring, the port frequencies must be set appropriately. In the port configuration table (Channel – [Channel Config] > "Port Config" > "Port Settings..." > "Arb Frequency"), the (source) frequency of Port 1 is set to the desired RF frequency (here: the channel base frequency/sweep range f_b). An external generator provides a fixed stimulus signal at 1 GHz. To measure the up-converted IF signal, the (receive) frequency at Port 2 is set to f_b + 1 GHz.



Independent source powers for Port 1 and Gen1 can be configured in addition, if so desired.

Internal Second Source

If an internal second source is available, the mixer measurements outlined above (and many other measurements) can be performed without an additional external generator; see Section 4.7.15, "Second internal generator", on page 258. However, be aware of the limited source connectivity (described there).

4.7.3.1 Calibration options

In arbitrary mode, the R&S ZNB automatically calibrates the source and receive frequency ranges of all ports, according to the frequency conversion settings in the "Port Settings" dialog or in dedicated configuration dialogs.

- SMARTerCal is the recommended calibration method for frequency conversion measurements.
- For measurements on non-linear DUTs, an additional scalar power calibration is recommended; for details refer to Section 4.5.7, "SMARTerCal", on page 193.
- A load match correction (optional) involves an additional reverse sweep. It can provide a significant improvement of the transmission S-parameter measurements if the load ports are poorly matched.

Select the appropriate correction type according to your accuracy and speed requirements. Notice that none of the correction types provide the phase information for transmission S-parameters.

4.7.3.2 Scalar mixer measurements

Scalar Mixer measurements are included in option R&S ZNB-K4.

RF mixers convert an RF signal at one frequency into a signal at another frequency. The frequency that is to be shifted is applied at the RF input. The frequency shifting signal (from a local oscillator, LO) is applied to the RF mixer's LO port, resulting in an output signal at the mixer's Intermediate Frequency (IF) port. For a given RF signal, an ideal mixer would produce only two IF outputs: one at the frequency sum of the RF and LO (IF = RF + LO), and another at the frequency difference between the RF and LO (IF = |RF – LO|). Filtering can be used to select one of these IF outputs and reject the unwanted one.

Meaning of S-parameters

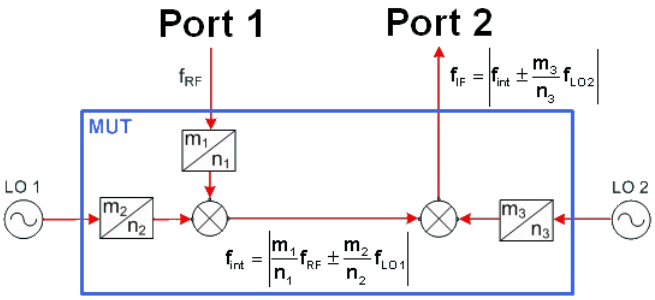
The frequency-converting property of the mixer (i.e. the fact that incident and transmitted waves are at different frequencies) causes a loss of phase information. While a scalar measurement is active, the reverse transmission parameter S_{12} is unavailable; the magnitude of the forward transmission parameter S_{21} describes the conversion gain. The conversion gain measurement can be improved by a source match correction (included in the SMARTerCal) and a load match correction (optional). The phase information, including the group delay, is meaningless.

In the scalar mixer mode the analyzer provides the following functionality:

- Configuration of the RF and LO signals and measurement of the generated IF signal.
- System error correction and power calibration of the signal sources and of the IF receiver. A SMARTerCal is recommended for this purpose; see Section 4.7.3.1, "Calibration options", on page 234.
- The mixer mode can be used to test important performance parameters of RF mixers such as frequency ranges, conversion loss, compression, and isolation.

Two-stage mixer measurements

The scalar mixer measurement is also suited for measuring a system of two mixers with frequency multipliers at their RF and LO inputs. The RF and LO input frequencies of the first mixer are both multiplied by an integer fraction; the converted output signal f_{int} is fed to the second mixer, together with the multiplied second LO signal. The analyzer measures the output signal of the second mixer at its IF input port. The general test setup is shown below; the mixer system under test (MUT) is enclosed in a blue rectangle.



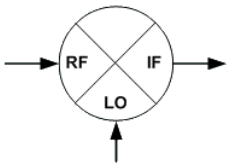
The figure above also shows the possible output frequencies of the two mixers. The actual values of f_{int} and f_{IF} depend on the RF and LO frequencies and of the measured conversion (lower sideband or upper sideband with up- or down-conversion). The analyzer automatically calculates all frequencies and sets its receiver according to the settings made.

A test setup with two mixers requires 3 independent source ports plus one receive port. An instrument with two internal sources requires one external generator, all other instruments require two external generators.

A standard mixer measurement with a single mixer stage and no frequency multipliers corresponds to the figure above with the second mixer and LO 2 omitted and $m_1 = n_1 = m_2 = n_2 = 1$.

Mixer diagrams

The mixer signal diagrams show the parameters of the mixer input signals (RF, LO) and of the mixing product (IF signal, output).

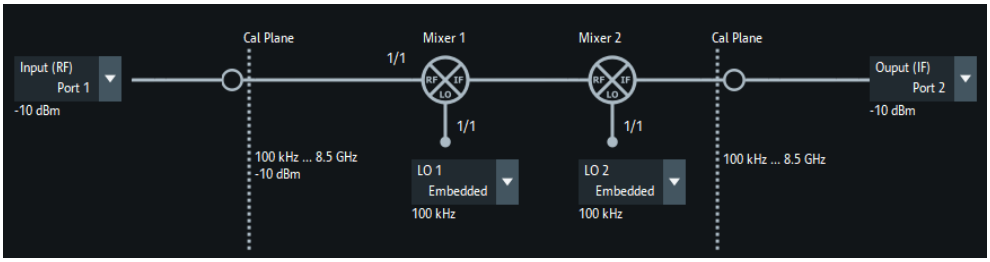


- The RF signal is the stimulus signal that the analyzer generates with the current channel settings. After a reset, the frequency and power of the RF signal is as defined in the Channel – Stimulus menu. The RF signal parameters can be changed in the "Power" and "Frequencies" dialogs.
- The Local Oscillator (LO) signal is an additional RF signal that is either generated by the network analyzer (at one of the ports that are not used for the RF and IF signals) or by an external generator. A test setup with two mixers involves two independent LO signals at both mixers.
- The IF signal is the mixer output signal (mixing product), which is at one of the following conversion frequencies: IF = LO + RF or IF = |LO – RF|, i.e. LO – RF (for LO > RF) or IF = RF – LO (for RF > LO). The IF frequency is selected in the "Frequencies" dialog. A test setup with two mixers involves two independent conversion settings.

The signal description above, with the swept RF signal and the LO signal at a fixed frequency, corresponds to the default configuration. In the "Frequencies" dialog, you can select any of the signals as a "Sweep/CW" signal. You can set the frequency range for this signal via "Start/Stop" or "CW Frequency". A second signal is at a "Fixed" frequency, and the third at the calculated sum or difference frequency ("Auto").

The labeling of the complete diagrams depends on the sweep type.

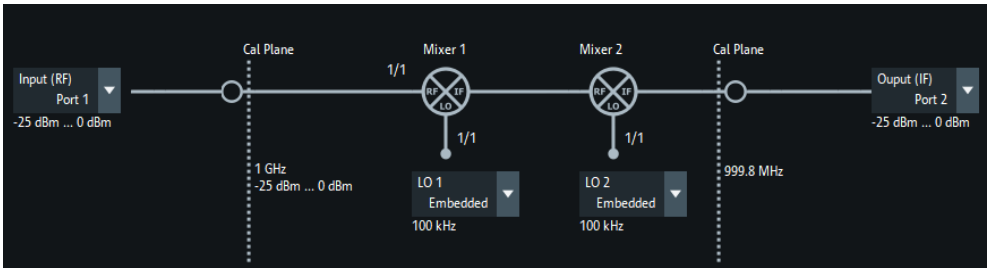
- The following mixer signal diagram corresponds to a **frequency sweep**:



  - RF signal (left side): Analyzer port number (e.g. Port 1), frequency sweep range (or fixed frequency, if the LO signal is swept), CW power, frequency conversion settings (1 / 1 denotes no conversion).
  - LO signals (1 or 2, from below): Signal source (analyzer port or external generator), fixed power and frequency (or frequency sweep range, if the RF signal is at fixed frequency), frequency conversion settings.

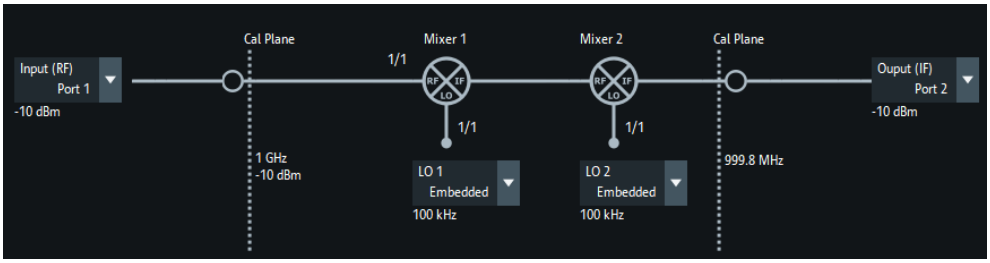
– IF signal (right side): Analyzer port number (e.g. Port 2), frequency range = (sweep range + LO) or |sweep range – LO|, expected fixed power.

- The following mixer signal diagram corresponds to a **power sweep**:



– RF signal (left side): Analyzer port number (e.g. Port 1), power sweep range or fixed power, CW frequency, frequency conversion settings (1 / 1 denotes no conversion).
– LO signals (1 or 2, from below): Signal source (analyzer port or external generator), fixed power or power sweep range, CW frequency, frequency conversion settings.
– IF signal (right side): Analyzer port number (e.g. Port 2), fixed frequency = (RF + LO) or |RF – LO|, expected power range.

- The following mixer signal diagram corresponds to a **Time or CW mode sweep**:



– RF signal (left side): Analyzer port number (e.g. Port 1), fixed power, CW frequency, frequency conversion settings (1 / 1 denotes no conversion).
– LO signals (1 or 2, from below): Signal source (analyzer port or external generator), fixed power, CW frequency, frequency conversion settings.
– IF signal (right side): Analyzer port number (e.g. Port 2), fixed frequency = (RF + LO) or |RF – LO|, expected fixed power.

The ports in the mixer signal diagrams are physical ports. To measure mixers with differential inputs, define a logical port configuration and enter one of the physical ports that belong to the logical port. The analyzer implicitly accounts for the logical port settings.

4.7.3.3 Intermodulation measurements

Option R&S ZNB-K14

The intermodulation measurement requires option R&S ZNB-K14 and, as a prerequisite, the Frequency conversion measurements option R&S ZNB-K4.

The "Intermodulation Wizard" facilitates the measurement configuration and the selection of results (see Section 5.2.3.1, "Intermodulation tab", on page 334).

If external switch matrices are used, intermodulation measurements are not supported.

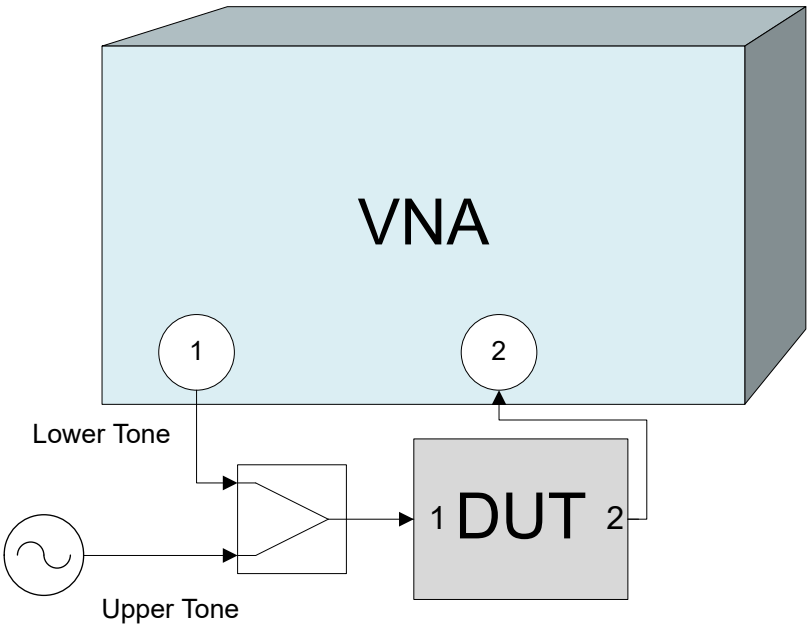
An intermodulation measurement is performed with two RF signals of equal power but different frequencies termed the upper and lower tone. The purpose of the measurement is to test the properties of a DUT that is supplied with a signal that covers a frequency band, typically a modulated RF channel. To simulate this scenario, the frequency difference ("tone distance") between the upper and lower signal is chosen to be small compared to the frequencies of the two tones:

$f_L \approx f_U$ or $\Delta f = f_U - f_L \ll f_L$

A nonlinear behavior of the DUT causes emissions at frequencies which correspond to sums and differences of the upper and lower tone frequencies and their integer multiples. These intermodulation products can be near the upper and lower tone frequencies, as long as their order is odd.

The analyzer measures the intermodulation products of k^{th} order IMk (where k = 3, 5, 7, 9) at the lower tone frequency minus (k – 1)/2 times the tone distance and at the upper tone frequency plus (k – 1)/2 times the tone distance; see also "Intermodulation measurement results" on page 240.

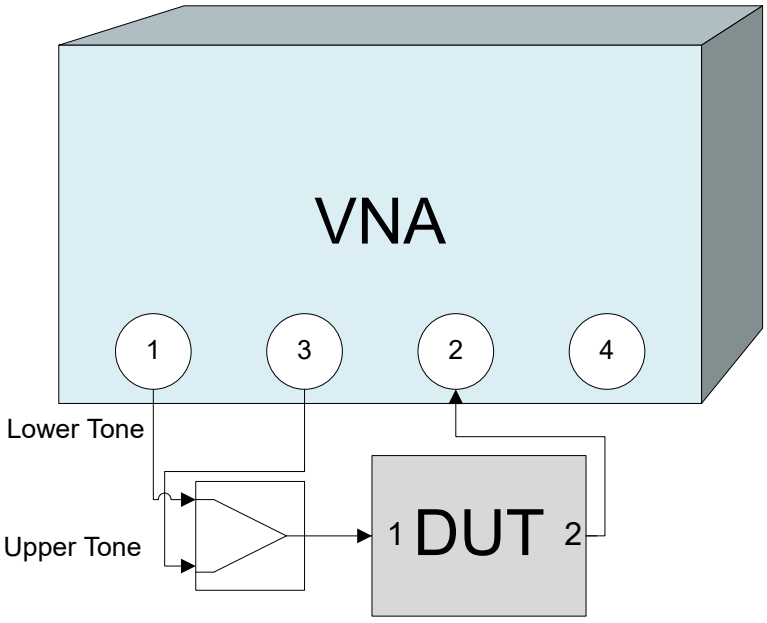
For a two-port R&S ZNB, a test setup with an external generator and a combiner is required.



The lower tone signal is generated at port 1, the upper tone is provided by the external generator. Both signals are combined externally and fed to the DUT input. The intermodulation quantities can be measured at the DUT output (transmitted wave b2). It is possible to interchange the ports no. 1 and 2.

For a R&S ZNB with two internal sources, you can use any combination of analyzer ports that are supplied by different internal sources to generate the lower and upper tones (see Section 4.7.15, "Second internal generator", on page 258).

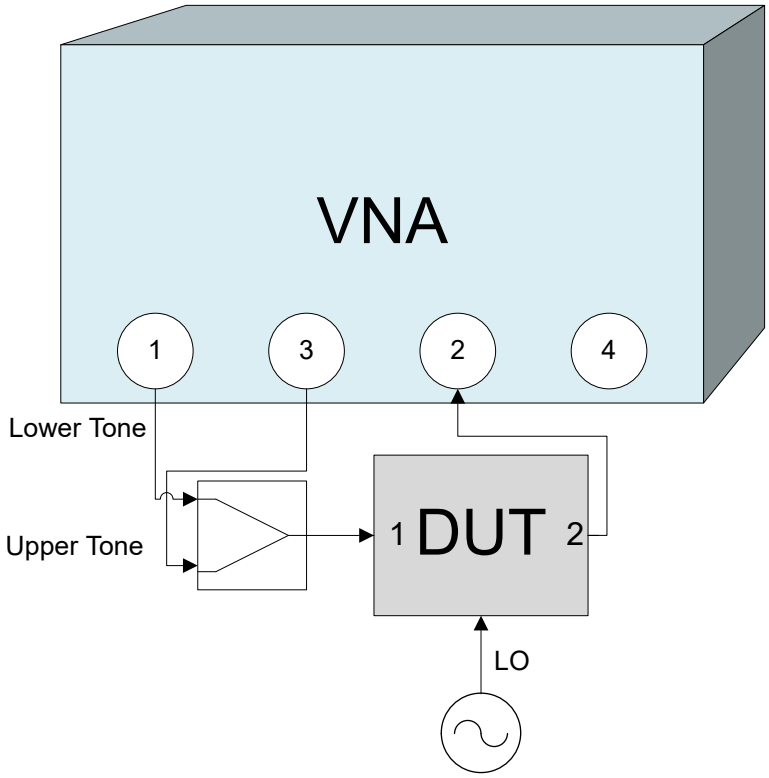
In the following example, port 3 of a R&S ZNB replaces the external generator.



An external combiner is required anyway.

Operation with more than one source is *not* supported, if External switch matrices are part of the RF connection configuration.

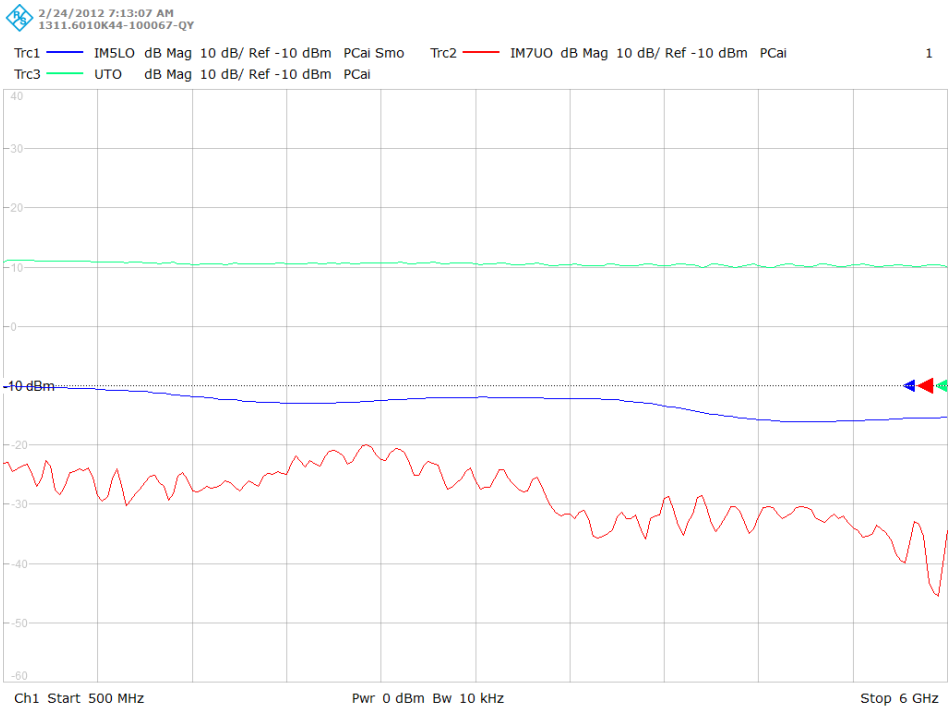
The intermodulation distortion measurement can be extended to frequency-converting DUTs. E.g., it is possible to feed the two-tone source signal to the RF input of a mixer and measure the intermodulation distortion of the IF output signal, after conversion with an additional LO signal.



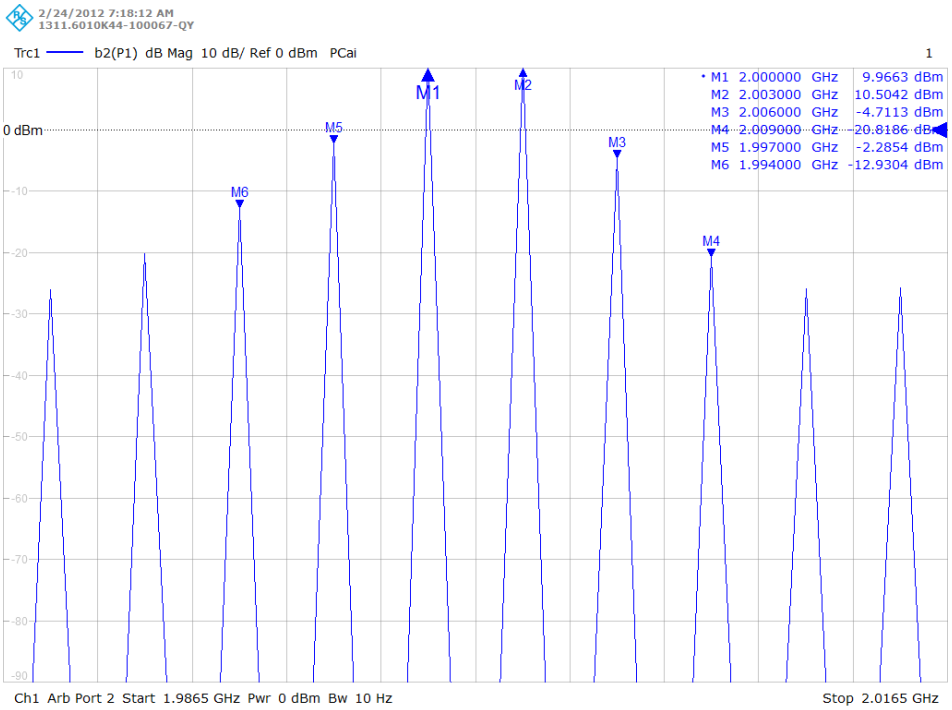
Intermodulation measurement results

The intermodulation measurement provides two different types of results:

- In the swept measurement, the analyzer performs a frequency or power sweep of the two-tone stimulus signal and displays the selected intermodulation quantities as a function of the lower-tone frequency or power.



- In the intermodulation spectrum measurement ("CW Mode Spectrum"), the frequency and power of the lower and upper tones is kept constant. The analyzer displays all intermodulation products near the signals up to a selectable order.



Intermodulation quantities

Intermodulation products

A nonlinear DUT that is supplied with a two-tone signal causes emissions at so-called **intermodulation products**, i.e. at sums and differences of the two frequencies and their integer multiples.

In general, with f_L denoting the lower and f_U the upper tone, intermodulation products of order *O* are defined as:

$k_1 \cdot f_L + k_2 \cdot f_U$, with $|k_1| + |k_2| = O$.

The analyzer can measure the power at intermodulation products of order k = 2, 3, 5, 7, 9, focusing on the "critical" intermodulation products closest to f_L and f_U.

With $\Delta f := f_U - f_L \ll f_L$, these intermodulation products are defined in the following table.

Table 4-25: Closest intermodulation products

Order	Lower	Upper
2	$\Delta f := f_U - f_L$	$f_U + f_L$
3	$f_L - \Delta f = 2\,f_L - f_U$	$f_U + \Delta f = 2\,f_U - f_L$
5	$f_L - 2\,\Delta f = 3\,f_L - 2\,f_U$	$f_U + 2\,\Delta f = 3\,f_U - 2\,f_L$
7	$f_L - 3\,\Delta f = 4\,f_L - 3\,f_U$	$f_U + 3\,\Delta f = 4\,f_U - 3\,f_L$
9	$f_L - 4\,\Delta f = 5\,f_L - 4\,f_U$	$f_U + 4\,\Delta f = 5\,f_U - 4\,f_L$

The R&S ZNB measures the intermodulation products *at the DUT output* (transmitted wave b_n). In the **non frequency-converting case**, it measures at the intermodulation products of the generated lower and upper tones.

In the **frequency-converting case**, the measured frequencies generally depend on where the non-linearity causing the intermodulation is located: at the RF or IF side of the DUT.

With the mixer function $MIX(f) = f + \Delta f_{MIX}$, the R&S ZNB (calibrates and) measures the intermodulation products of **order 2** at the IF frequencies defined in the following table.

Table 4-26: Mixer intermodulation measurements: Intermodulation products of order 2

Intermodulation location	Upper	Lower
DUT input or RF side	$MIX(f_U - f_L) = f_U - f_L + \Delta f_{MIX}$	$MIX(f_U + f_L) = f_U + f_L + \Delta f_{MIX}$
DUT output or IF side	$MIX(f_U) - MIX(f_L) = f_U - f_L$	$MIX(f_U) + MIX(f_L) = f_U + f_L + 2\,\Delta f_{MIX}$

For intermodulation products of **odd order** $O \geq 3$, the frequency shifts of the mixers cancel. Writing

$$\mathrm{IP}_O(f_1, f_2) = \frac{O+1}{2} f_1 - \frac{O-1}{2} f_2$$

for the intermodulation products of order *O*, we have

$$\begin{aligned}\mathrm{IP}_O(\mathrm{MIX}(f_1), \mathrm{MIX}(f_2)) &= \frac{O+1}{2}(f_1 + \Delta f_{MIX}) - \frac{O-1}{2}(f_2 + \Delta f_{MIX}) \\ &= \frac{O+1}{2} f_1 - \frac{O-1}{2} f_2 + \Delta f_{MIX} \\ &= \mathrm{MIX}(\mathrm{IP}_O(f1, f2))\end{aligned}$$

and hence a distinction between "before mixer" and "after mixer" is not required.

Intermodulation suppression and intercept points

The **intermodulation suppression** IMk_{rel} is the ratio of the power of an intermodulation product to the power of the lower tone fundamental wave in dB, measured at the DUT output.

For each intermodulation product, an output **intercept point** IPk_{out} is defined as follows:

$IPk_{out} = P_{L, out} – IMk_{rel} / (k – 1)$

In this formula, $P_{L, out}$ denotes the lower tone power in dBm, measured at the DUT output. In analogy, the input intercept point IPk_{in} is defined as:

$IPk_{in} = P_{L, in} – IMk_{rel} / (k – 1)$

The intermodulation suppression generally decreases with increasing stimulus power. The intercept point is equal to the lower tone power for which the intermodulation suppression would reach 0 dB. Output and input intercept point differ by the attenuation of the lower tone signal upon transmission through the DUT. The intercept point is a mathematical concept. For most DUTs, it is beyond the nominal power operating range.

Filter settings

Intermodulation measurements require frequency-selective filter settings. When an intermodulation result is selected, the analyzer checks whether the IF bandwidth of the active channel is 1/100 of the distance between the upper and the lower tone ("Tone Distance") or less. If not, the analyzer displays a tooltip.

Selectivity is automatically set to high, but can be changed manually.

4.7.4 Receiver bandwidth 10 MHz

Option R&S ZNB-K17 / R&S ZNBT-K17

This software option increases the maximum receiver bandwidth of a R&S ZNB from 1 MHz to 10 MHz.

4.7.5 Frequency resolution 1 mHz

Option R&S ZNB-K19

This software option improves the frequency resolution of a R&S ZNB to 1 mHz.

4.7.6 Noise figure measurement

Option R&S ZNB-K30

Software option R&S ZNB-K30 allows you to measure the noise figure of a 2-port DUT, operating in its linear range.

4.7.6.1 Introduction

Noise figure (or *noise factor*, if the linear representation is chosen) is an important figure of merit for system designers and eventually determines the overall system performance of a receiver.

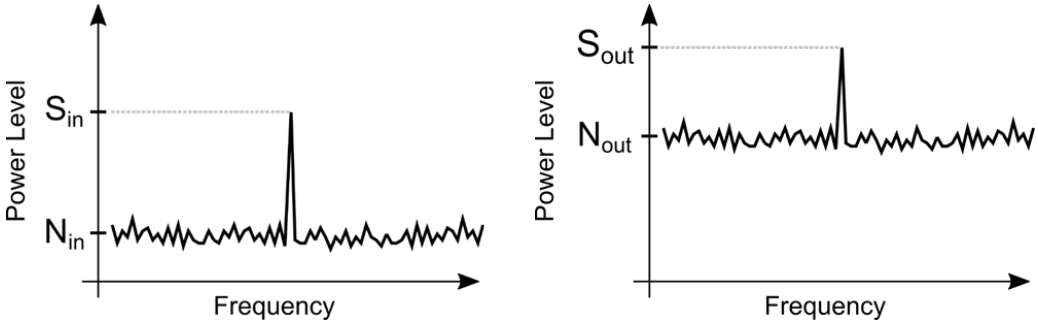


Figure 4-24: Exemplary spectrum for a single tone excitation (left = DUT input; right = DUT output)

The general definition of the noise factor F_D is the ratio of the signal to noise ratio (SNR) at the input, to the SNR at the output of a DUT. In other words, it describes the degradation of the SNR from the input to the output of a DUT.

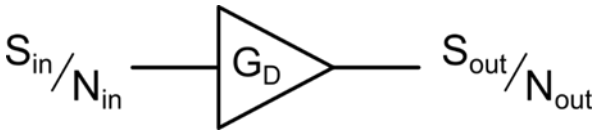


Figure 4-25: Schematic showing the signal and noise powers relevant for noise figure calculations

Based on this formulation the equation can be written in linear terms as:

$$F_D = \frac{SNR_{in}}{SNR_{out}} = \frac{S_{in}/N_{in}}{S_{out}/N_{out}} = \frac{S_{in}N_{out}}{S_{out}N_{in}} = \frac{N_{out}}{G_D N_{in}}$$

S_{in} and S_{Out} denote the signal power at the input and output of the DUT, respectively. N_{in} and N_{out} represent the noise present at the input of the DUT and at the output of the DUT, respectively. The gain G_D of the DUT is defined by the ratio of the output signal power to the input signal power:

$$G_D = \frac{S_{out}}{S_{in}}$$

The resulting formula for the noise factor F_D shows that the calculation depends on precise knowledge of:

- the noise that is present at the input of the DUT,
- the noise that is present at the output of the DUT
- the gain of the DUT

These quantities must be measured during the different steps of a noise figure measurement. In a post-processing step, these individual measurements are then used to calculate the noise figure of the DUT.

Typically, the noise factor F_D is displayed in decibel which is then called the noise figure $NF_D = 10 \log F_D$. Noise figure and noise factor are used interchangeably throughout this manual.

4.7.6.2 Algorithmic implementation

Cold source method

Historically, the noise figure of DUTs was measured using the Y-factor method with a noise figure analyzer or with a spectrum analyzer. Because these measurement instruments had no internal sources, they used an external broadband noise source (switched on and off) to measure the noise and the gain of the DUT.

Nowadays, with the use of modern network analyzers, there is no need for an additional noise source and the Y-factor method. Instead, the so-called cold source method is utilized, which uses broadly available and highly accurate power meters for calibration. The noise emitting from the DUT is measured with switched off internal VNA source and a matched termination at the DUT input (typically a VNA port). The gain of the DUT is measured with an active source and the calibrated VNA receivers.

Equivalent circuit

The internal calculations are performed in the noise temperature representation.

Compared to calculations using noise factors (or noise figures), this method can easily account for different physical temperatures of the involved components. Furthermore, with noise temperatures there is no need to apply corrections factors, if the components are on a different temperature than the reference temperature T_0 (= 290 K). The equivalent circuit, which is internally used for the calculation of the DUT noise factor F_D, is shown in Figure 4-26.

G_{ext} comprises the total external loss (net gain) between the R&S ZNB port and the calibration plane, including connecting cables, external source attenuator, additional external attenuators, adapters, etc.

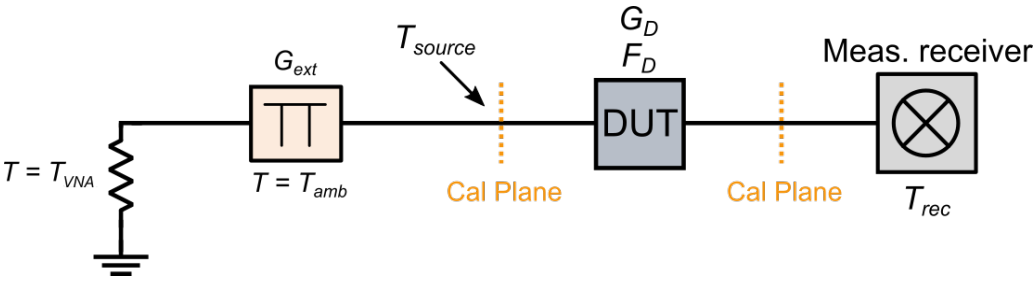


Figure 4-26: Equivalent circuit used for the calculation of the DUT noise factor

A noise temperature T is converted into the corresponding noise factor F using the equation:

$$F = 1 + \frac{T}{T_0}$$

Source noise temperature

As already mentioned, it is important to have precise knowledge of the noise present at the input of the DUT.

The input noise temperature T_{Source} is mainly dominated by the external source attenuator. Since it is required to use attenuations ≥ 40 dB, the noise from the VNA can be neglected. The temperature of the external source attenuator and other additional attenuators is given by the ambient temperature.

The ambient temperature is by default chosen to 296 kelvins, which is close to a typical laboratory temperature of 23 °C. However, for precise measurements the ambient temperature can be adjusted to the actual room temperature. See "Amb Temp in Kelvin" on page 354.

Due to the required values for the external source attenuator, the “hot” internal temperature of the R&S ZNB is suppressed and an effective noise temperature $T_{Source} = T_{amb}$ is assumed at DUT input.

Receiver noise temperature

The noise that is present at the output of the DUT cannot be measured directly. In fact, the measured noise is always the sum of the noise emitting from the DUT and the noise present in the receiver. An additional step is required during calibration to characterize the noise of the receiver.

For the receiver NF calibration, a well matched 50 Ω load connected to the receiving port and the noise power is measured. The model used for the receiver noise calibration is shown in Figure 4-27 and depicts the matched load (at ambient temperature) directly connected to the receiver.

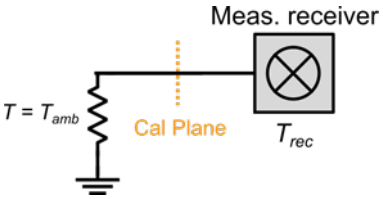


Figure 4-27: Model used for the receiver noise calibration

Using the model above, it is possible to calculate the receiver noise temperature T_{rec} using the following formula:

$$T_{rec} = \frac{N_{M,rec}}{kB} - T_{amb}$$

In this formula, $N_{M,rec}$ is the measured noise power during calibration, k is the Boltzmann constant (1.38064852e-23 J/K), and B is the equivalent noise bandwidth of the used IF filter.

The equivalent noise bandwidth represents the cumulated normalized filter response. In other words, it is the bandwidth of a rectangular filter which has the same integrated noise power as the real filter. The calculation is done by the following equation:

$$B = \frac{1}{2\pi}\int_0^{\infty}\left|\frac{H(j\omega)}{H_{max}}\right| d\omega$$

where $H(j\omega)$ is the magnitude of the IF filter, H_{max} is the maximum value of $H(j\omega)$.

DUT noise figure

The calculation of the noise figure (NF) of a non-frequency-converting DUT requires a calibrated NF channel in the R&S ZNB setup. During calibration, important system parameters such as the Receiver noise temperature are determined. The final noise factor F_D is then calculated using the following equation:

$$F_D = \frac{\frac{N_{DUT,meas}}{kB} - T_{rec}}{G_D T_0} - \frac{T_{source} - T_0}{T_0}$$

where

- $N_{DUT,meas}$ is the noise power measured with the DUT connected to the VNA
- T_{rec} and T_{source} are the source and receiver noise temperature, respectively, determined during calibration
- G_D is the measured gain of the DUT
- T_0= 290 K is the reference temperature

Harmonic correction

Typically, VNA receivers do not have any preselection filters within their signal paths. However, due to their architecture, they exhibit additional receiving windows at higher-

order harmonics of the local oscillator frequency. Frequencies in these higher-order sidebands are down-converted to the same IF frequency as the intended RF frequency. Without additional measurements, these sideband contributions cannot be separated from the contribution of the intended RF frequency band.

For typical network analyzer measurements with an active driving signal (S-parameters, intermodulation, compression, ...), unsuppressed higher-order sidebands are not an issue. Also for noise figure measurements on narrowband devices with less than an octave bandwidth and an out-of-band suppression of more than 30 dB, they do not impose significant measurement errors. But for noise figure measurements on a broadband DUT with several octaves of bandwidth (e.g. a distributed amplifier), they certainly cannot be ignored.

Noise figure option R&S ZNB-K30 uses a patented algorithm to remove the error introduced by the higher-order sidebands, and to calculate the true NF of a DUT. Based on additional measurements during calibration and measurement phase, the imperfect receiver response is corrected mathematically.

Noise power spectral density

The noise power spectral density, or noise density for short, is calculated as

$NPSD = ND = N_{meas,\ corr} / B$

where $N_{meas,\ corr}$ denotes the measured sideband-corrected noise power, and B the equivalent noise bandwidth (see "Receiver noise temperature" on page 246). $N_{meas,\ corr}$ is basically independent of what is currently connected to the test ports, so no distinction between calibration and DUT is required.

4.7.7 Measurement uncertainty analysis

Option R&S ZNB-K50/R&S ZNB-K50P

Measurement uncertainty analysis is available if option R&S ZNB-K50 and METAS VNA Tools (version 2.2.6 or higher) are installed on the instrument.



Option R&S ZNB-K50**P** offers the same functionality as option R&S ZNB-K50, but comes with METAS VNA Tools **p**reinstalled. It can be shipped with new instruments or installed at Rohde&Schwarz service.

The purpose of the METAS VNA Tools software is to compute uncertainties of S-parameter measurements. After registration, you can download it from the VNA Tools Internet site (https://www.metas.ch/vnatools).

- The uncertainty analysis of the METAS VNA Tools is limited to S-parameter measurements. Uncertainty analysis of wave quantities is not supported.
- The uncertainty calculations are in line with EURAMET Calibration Guide No. 12 Version 3.0 (03/2018).
- If external switch matrices are used, measurement uncertainty analysis is **not supported**.

METAS Calibration

The VNA firmware directly interacts with the DLLs of the METAS VNA Tools. During a "METAS Calibration", the VNA firmware collects all data that are required to set up a METAS VNA Tools project. After the "METAS Calibration"METAS calibration is completed, the analyzer is ready for real-time uncertainty analysis: after each sweep it passes the raw (uncalibrated) S-parameters to the VNA tools and receives the calibrated S-parameters plus uncertainties (95% confidence ellipse in the complex plane) in return.

The calibrated S-parameter traces plus their uncertainty bands can be displayed in magnitude, phase, real and imaginary trace formats. The display of uncertainties in complex diagrams (polar, Smith) is not supported.

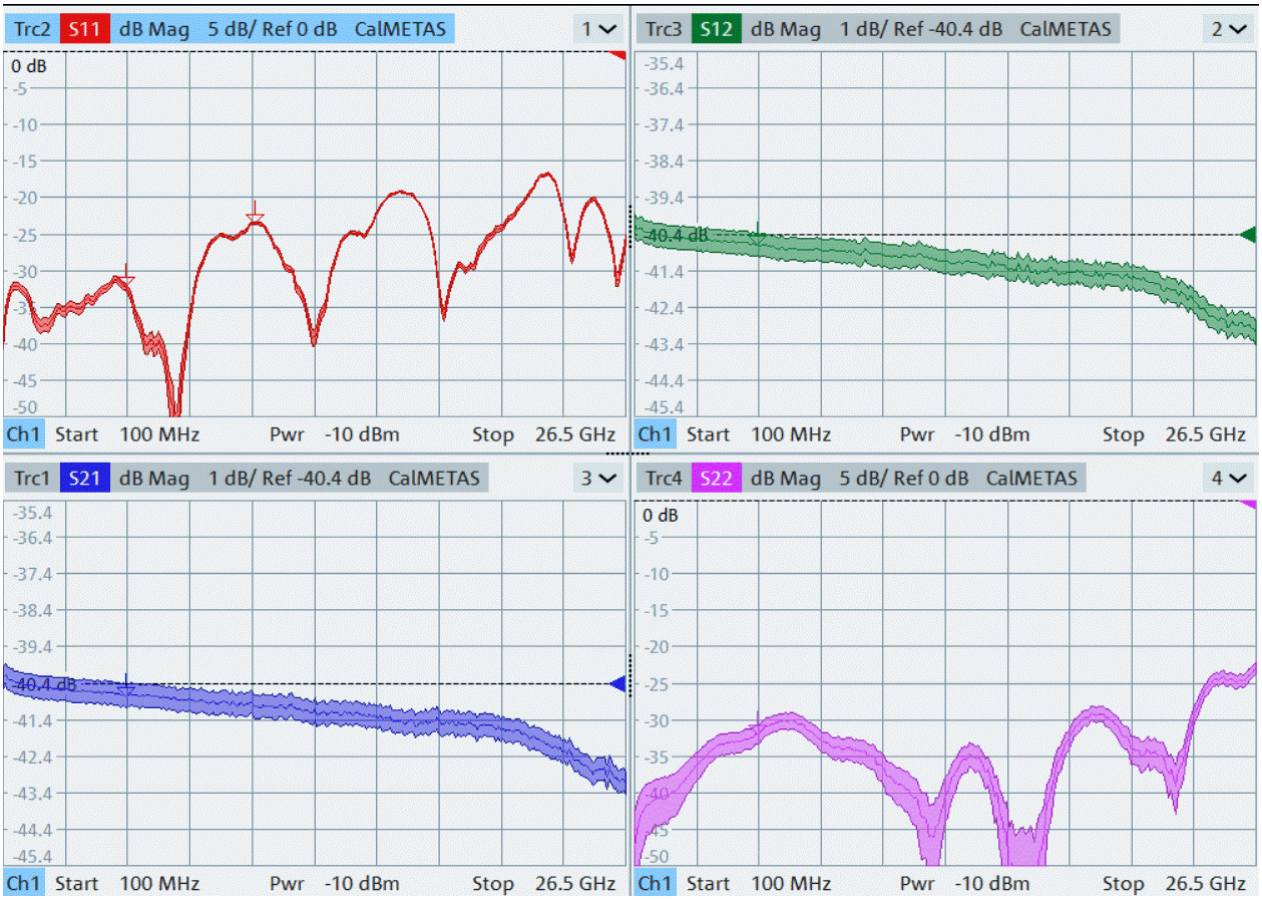


Figure 4-28: S-parameter measurements with uncertainty bands

Required input data for METAS VNA Tools:

- METAS VNA model (noise floor, trace noise, receiver linearity, drift)
  For R&S ZNB variants that support option R&S ZNB-K50, the VNA model is provided by the analyzer firmware.
- Connector types
  Part of the uncertainty calculation is the number of reconnections between calibrated ports and calibration standards or DUT. METAS VNA Tools provides "repeatability models" for various connector types.
- METAS cable model
  As cables (and cable movements) contribute to the measurement uncertainty, the magnitude and phase stability of the cables must be considered. The VNA firmware uses the cable models available in the local METAS VNA Tools installation. You can add your cable models using its graphical user interface. Cable models that ship with the VNA firmware are automatically added to the METAS database.
- Calibration standard characterizations with uncertainties
  Standard cal kit data does not contain information about the uncertainty of the characterization. However, to get meaningful results, these uncertainties must be considered and hence a "characterization with uncertainties" must be used. Rohde&Schwarz calibration kits with accredited calibrations (ACA) include uncertainties.
- Channel setup (IF bandwidth, power, ...) and calibration data
  Required to calculate the calibrated S-parameters and their uncertainties. If relevant settings are changed on the VNA, these changes are also propagated to the METAS VNA Tools.
- Measured S-parameters (uncalibrated)
  The raw ("uncalibrated") data are handed over to the METAS VNA Tools after every sweep for real-time correction and uncertainty analysis.

Only a single VNA channel can be METAS calibrated at a time.

Verification

To get precise information about the absolute measurement uncertainty after calibration, you can use verification standards, which are DUTs precisely characterized by the manufacturer. Verification kits, which include multiple standards, deliver the most accurate results for the measurement uncertainty.

Verification kits R&S ZV-Z4xx from Rohde&Schwarz contain a male and a female offset short, a male and a female mismatch, an attenuator and a stepped through. These verification standards differ substantially from the conventional calibration standards – Open, Short and Match – because they have different impedances. They are characterized in steps of 250 MHz and specified together with their measurement uncertainties. Characterization and uncertainties are accredited by the DAkks (Germany's national accreditation body).

The VNA firmware compares the uncertainties calculated by the METAS VNA Tools with the accredited uncertainties and presents both uncertainty bands graphically. To

pass verification, the calculated band must be within the accredited band for the specific standard.

Cal kits and verification kits with uncertainties

Cal kits and verification kits with uncertainties have to be provided as `*.zip` or `*.scolv` files. The comprised standards must be provided as `*.sdatb`, `*.sdatcv`, or `*.calstd` files. The standard filenames must convey information about the file content:

- Standard type
  - Cal kit: `open`, `short`, `load`, `match`, `through`, `sliding load`, or `sliding match`
  - Verification kit: `offset short`, `mismatch`, `airline`, `mismatch airline`, `att`, `attenuator`, or `stepped thru`
- Gender: `(f)`, `(m)`, `_f_`, `_m_`, `female`, `male`, `(fm)`, `_fm_`, ...
  If gender information is not found, the standard is assumed sexless.

4.7.8 SNP assistant

R&S ZNB-K100

The SNP Assistant (SNPA) makes the S-parameter characterization of multiport DUTs a breeze. Once you have declared the topology of your DUT and the sweep to be performed, the SNPA can initialize the DUT's S-parameters with plausible "idealized" values. These values can then be replaced by measured and/or imported traces (Touchstone data).

The SNPA's current data (DUT topology, sweep properties, S-parameter traces) can be persisted to an SNPA project file (`*.snp`), and recalled at a later point in time.

Both Touchstone import and S-parameter measurements are guided processes. The proposed measurement steps are optimized w.r.t. the required number of cable changes.

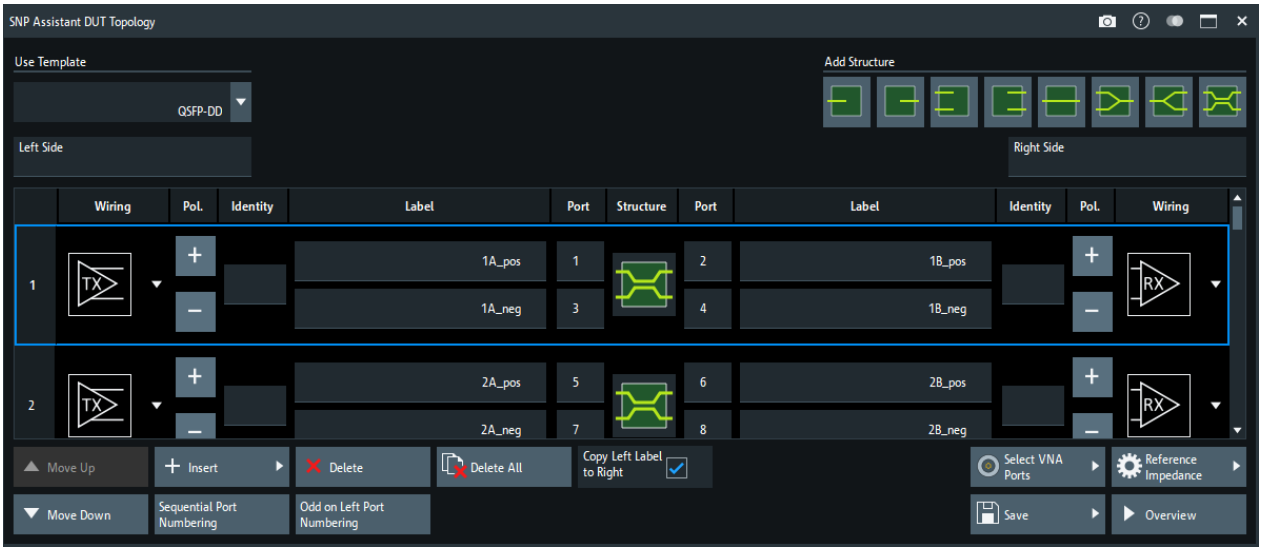


Figure 4-29: Step 1: Declare the DUT topology

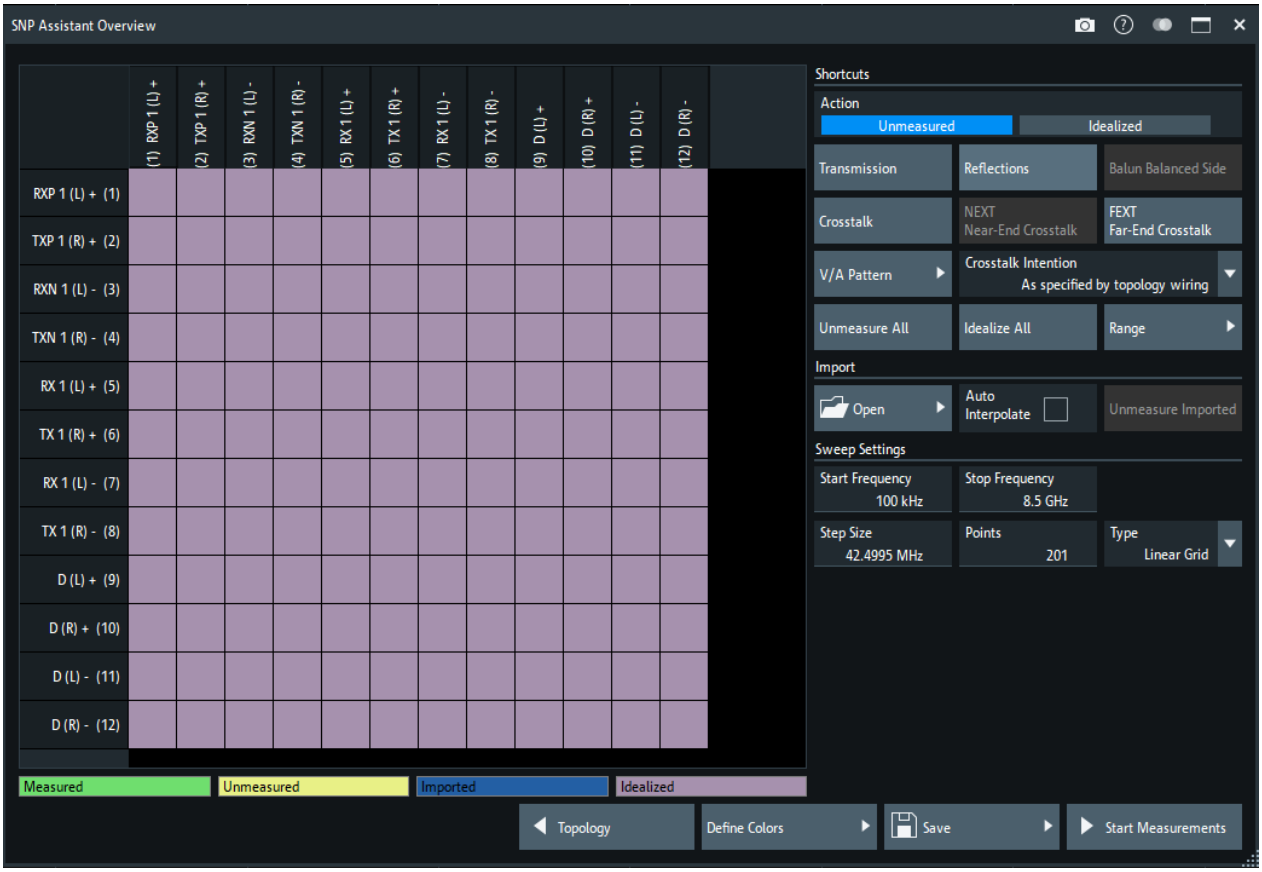


Figure 4-30: Step 2: Specify S-parameters to be measured/imported

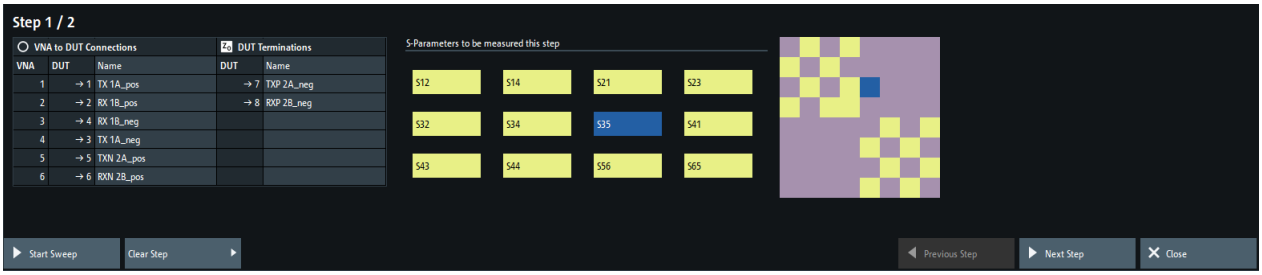


Figure 4-31: Step 3: Measure and/or import S-parameter data

4.7.9 Eazy de-embedding based on IEEE 370

Option R&S ZNB-K210

EaZy deembedding (EZD) is an implementation of the IEEE algorithm for generating deembedding files from a symmetrical 2x Thru measurement.

Option R&S ZNB-K210 allows you to model a test fixture using the EZD fixture modeling tool and to deembed selected ports using the generated touchstone files.

The EZD fixture modeling proceeds in the following steps:

1. Measure one or more test coupons for the related fixture.
2. Run the EZD tool to calculate the Touchstone files modeling the test fixture.

   The calculated `sNp` files can then be used to deembed the DUT at selected ports.

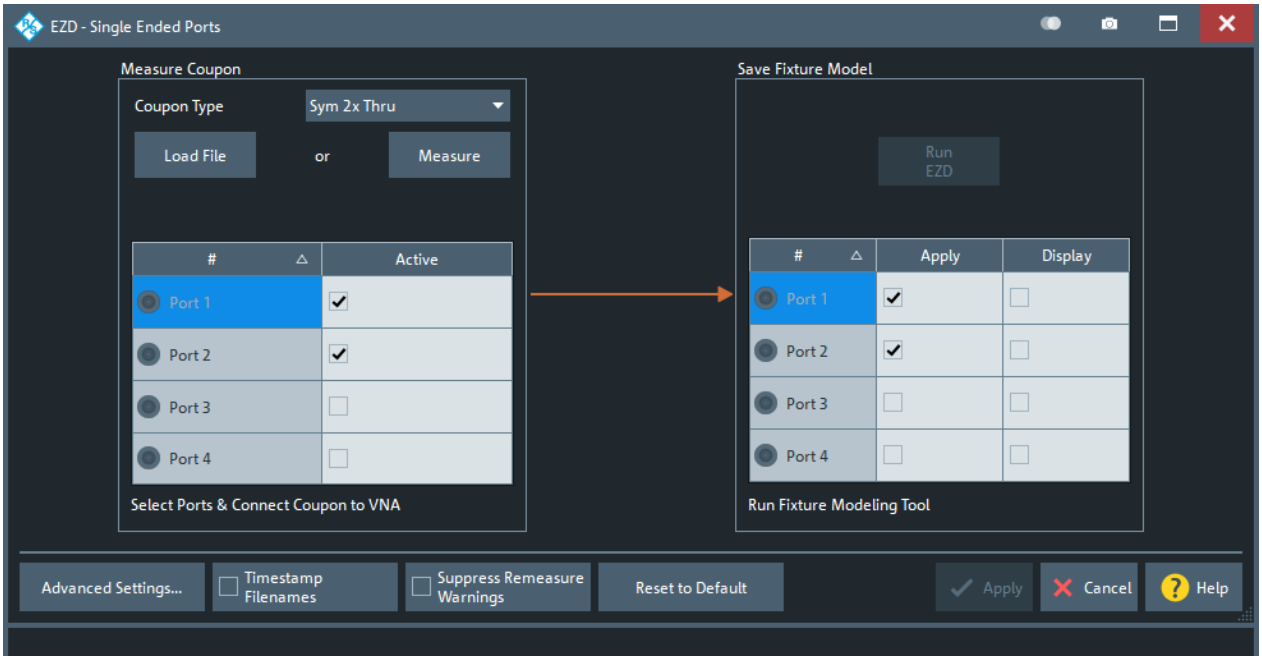


Figure 4-32: Fixture modeling: EZD – Single Ended Ports dialog

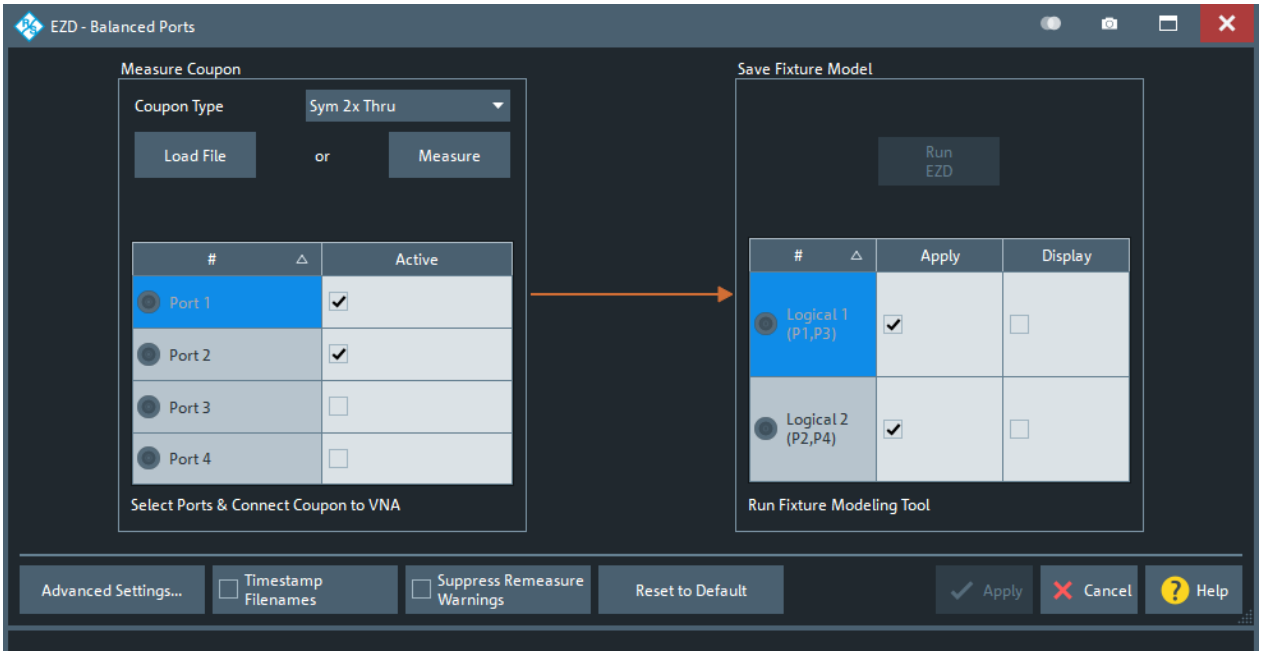


Figure 4-33: Fixture modeling: EZD – Balanced Ports dialog

- For general information about fixture de-embedding, see Section 4.6.2.9, "Fixture modeling and deembedding", on page 212.
- Option R&S ZNB-K210 can be ordered with a new instrument or retrofitted at Rohde&Schwarz service.

4.7.10 In-situ de-embedding

Option R&S ZNB-K220

This option offers a pre-installed or service-retrofitted version of AtaiTec's *In Situ De-Embedding* (ISD) tool for fixture de-embedding.

The tool is integrated into the de-embedding functionality of the analyzer firmware.

- For general information about fixture de-embedding, see Section 4.6.2.9, "Fixture modeling and deembedding", on page 212.
- For detailed information about the ISD tool, see the product pages at http://atai-tec.com/products/isd/

Tips for using ISD

The ISD User's Guide offers the following guidelines to get the most out of ISD:

1. The Touchstone files must have enough number of points.
   The minimum required is at least two points for every 90 degree phase angle change in insertion loss (or 8 points per wavelength).
2. The "2x Thru's" return loss must be smaller than its insertion loss.
3. The "2x Thru's" insertion loss must be relatively linear, with a few wavelengths at the highest frequency.
4. Use a 4-port "2x Thru" if strong far-end crosstalk (FEXT) is present.
5. Use a *complete* DUT w. fixture file for extraction.

   Source: "In Situ De-Embedding (ISD) User's Guide" – November 11, 2018

4.7.11 Smart fixture de-embedding

Option R&S ZNB-K230

This option offers a pre-installed or service-retrofitted version of PacketMicro's *Smart Fixture De-embedding* (SFD) for fixture de-embedding.

The tool is integrated into the de-embedding functionality of the analyzer firmware.

- For general information about fixture de-embedding, see Section 4.6.2.9, "Fixture modeling and deembedding", on page 212.
- For detailed information about the SFD tool, see the product pages at https://www.packetmicro.com/Products/sfd-tool.html.

4.7.12 Delta-L 4.0 PCB characterization

Option R&S ZNB-K231

Delta-L is a de-embedding methodology developed by Intel Corporation and is used for de-embedding strip lines that include vias. Delta-L calculates the loss of PCB interconnects and can be performed on single-ended or balanced interconnects.

The objective of Delta-L is to determine an insertion loss measurement defined on two or more coupon lines of differing lengths. This ratio of insertion loss in dB/inch can be applied to any trace length with similar vias.

For background information, see

- https://www.intel.com/content/dam/www/public/us/en/documents/guides/electrical-character-design-meth-guide-337658-rev001.pdf
- https://ieeexplore.ieee.org/document/7048423

The R&S ZNB uses the Clear Signal Solutions AITT-DL implementation of the Delta-L v4.0 algorithms (see https://clearsig.com/clearsig/tools/#Delta). It allows you to perform:

- 1L (one line/length) analysis
  Typically used for high-volume manufacturing (relative comparison)
- 2L analysis
  Typically used for board-quality validation (removes the test fixture impact)
- 3L analysis
  Typically used for material characterization (best accuracy, self-validation of results).

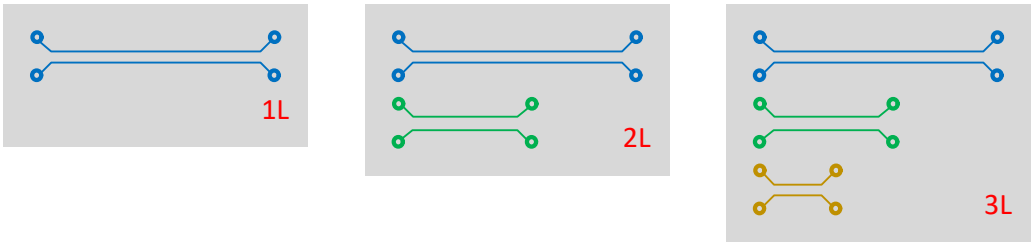


Figure 4-34: Test coupons for 1L, 2L and 3L measurements

Option R&S ZNB-K231 can be ordered with a new instrument or retrofitted at Rohde & Schwarz service.

4.7.13 Health and utilization monitoring service (HUMS)

Option R&S ZNB-K980

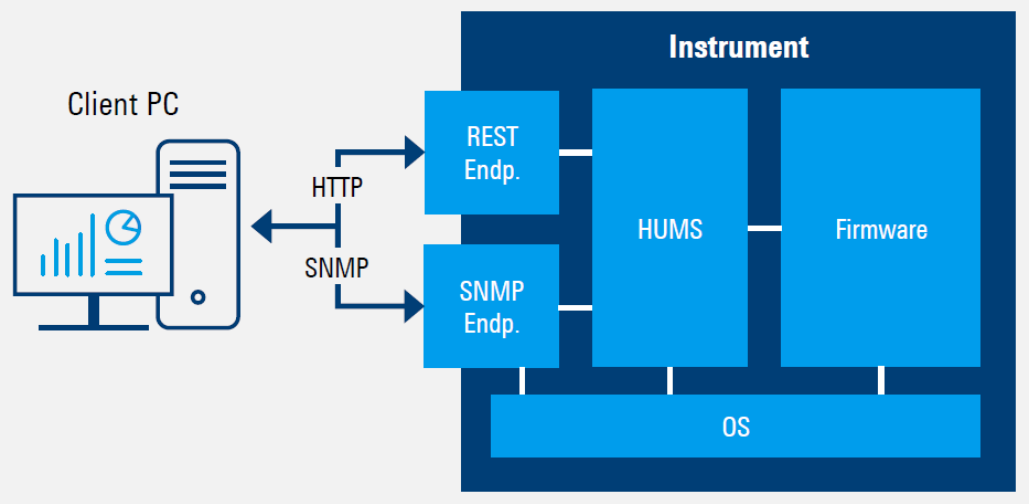
The R&S ZNB comes with a health and utilization monitoring system (HUMS) providing information about the R&S ZNB. Aim is to increase the overall utilization, to avoid downtime and to increase the overall security level of a fleet of instruments.

HUMS provides, for example, information about:

- Health status
- Characteristics (device footprint)
- Operation system / security patches
- Option usage
- Calibration and service information

Interfaces and protocols

The HUMS installation on the R&S ZNB includes an SNMP agent and a REST service with HTTP endpoints. So you can access the health and utilization information via LAN, using the SNMP protocol or the REST protocol. Accessing the data does not interfere with remote control via SCPI commands or with measurement execution.



Reference information for both protocols is available on the R&S ZNB at the address *http://<instrument>/api/hums/v1/documents?name=<interface>*.

For *<instrument>*, enter the hostname or the IP address of your instrument, as for access to the GUI.

For *<interface>* = *snmp*, you get a `.zip` file containing the MIB files for SNMP. For *<interface>* = *rest*, you get a web page with the OpenAPI specification of the REST API.

For further information about the HUMS service itself, see the R&S HUMS user manual.

4.7.14 Bias tees

Option R&S ZNB-B1 for R&S ZNB4|8

Bias tees are used to apply external DC voltages (biases) to the RF ports of a R&S ZNB. They are provided as additional rear panel BNC inputs (labeled PORT BIAS 1, 2,...).

- Availability:
  - For R&S ZNB and R&S ZNB8, bias tees are provided as an option (see below).
  - For 2nd generation R&S ZNB20 and for 1st generation R&S ZNB40 that are equipped with the new reflectometer board with ESD limiters, bias tees are **not** available.
  - For all other R&S ZNB models, they always equipped.
- Option R&S ZNB-B1 can only be equipped ex factory, and can only be ordered with new R&S ZNB4|8.
- Bias tees have an impact on the start frequency of the instrument. For details, refer to the data sheet and to [SENSe:]FREQuency... commands.
- There is no additional HW or SW switch for the bias tees. The voltage is applied to the RF port when you connect the voltage source to the corresponding PORT BIAS connector.

Hardware option R&S ZNB-B1 provides:

- Bias tees for two-port R&S ZNB4|8, order no. 1316.1700.02
- Bias tees for four-port R&S ZNB4|8, order no. 1316.1700.04

4.7.15 Second internal generator

Options R&S ZNB-B2 and R&S ZNB<20|26|40|43>-B2

Hardware options

- R&S ZNB-B2 "Internal Second Source for R&S ZNB4 and R&S ZNB8 (4-Port)"
- R&S ZNB<20|26|40|43>-B2, "Second Internal Generator for R&S ZNB<20|26|40|43> (4-Port)"

expand four-port network analyzers R&S ZNB with a second internal generator, which can be used as an additional stimulus signal, e.g for mixer measurements; see Section 4.7.3, "Frequency conversion measurements", on page 233.

Limitations

With two internal sources, the physical VNA ports are split into groups P_1 and P_2 such that source 1 can only drive ports in P_1 and source 2 can only drive ports in P_2:

- P_1 = {1,2} and P_2 = {3,4} for four-port R&S ZNB
- P_1 = {1,...,8} and P_2 = {9,...,24} for the R&S ZNBT

Independent signals require the source ports to reside in different port groups. On the R&S ZNB, for example, you cannot use ports 1 and 2 as independent source ports. A R&S ZNBT on the other hand must be equipped with at least 12 ports to allow independent signals.

4.7.16 Precision frequency reference

Option R&S ZNB-B4

An optional oven-controlled crystal oscillator (OCXO) improves the static frequency accuracy of the R&S ZNB. For details, refer to the data sheet.

4.7.17 GPIB interface

Option R&S ZNB-B10

This hardware option provides a GPIB bus connector according to standard IEEE 488 / IEC 625 (GPIB). The GPIB bus interface can be used for controlling the R&S ZNB from an external controller.

For interface details, refer to Section 13.3.3, "GPIB interface", on page 1704.

Remote control of external devices via GPIB

The current hardware of the GPIB interface does not support GPIB bus controller mode, i.e. it cannot be used to control external devices via GPIB.

If you want to use GPIB to control external devices from a R&S ZNB, you have to use an additional USB-to-IEC/IEEE adapter.

4.7.18 Device Control

Option R&S ZNB-B12

This option provides a PCIe and a Direct Control interface, both intended to control external devices.

- The PCIe interface is suitable for certain Rohde & Schwarz Signal Generators.

- The Direct Control interface is a proprietary interface for Rohde & Schwarz VNA extensions such as the external RFFE GPIO interface R&S ZN-Z15 or External switch matrices R&S ZN-Z8x.
  A suitable cable is available with option R&S ZN-B121.

Direct control of Switch Matrices

The Direct Control interface allows you to establish a direct connection between the VNA measurement bus and the FPGA controlling the routes of a switch matrix R&S ZN-Z8x. This bypasses the matrices' microcontroller and significantly reduces the switching times compared to management via USB or LAN.

4.7.19 Handler I/O (universal interface)

Option R&S ZN-B14

A network analyzer that is equipped with a Handler I/O (Universal Interface), can interact with an external part handler. The digital control signals on the interface connector indicate the possible start and the end of a measurement, as well as a global limit check result. Typically, the handler will insert the device to be tested into a test fixture and provide a trigger pulse to initiate the measurement. After the measurement is complete, the handler will remove and replace the device and sort it into pass/fail bins. For details, refer to Section 13.3.4, "Handler I/O (universal interface)", on page 1707.

4.7.20 RFFE GPIO interface

Internal: Option R&S ZN-B15 (1st generation R&S ZNB only)

External: Option R&S ZN-Z15

The trend in mobile radio communication is towards higher scale of integration of external components such as filters, switches, low noise and power amplifiers. The requirement for test and measurement equipment is to make it possible to control those components directly from the instrument without the need of additional external tools, e.g. a USB to RFFE adapter.

For these kinds of applications, Rohde & Schwarz has developed an RFFE-GPIO extension board for the R&S ZNB. This board is equipped with a 25-pin female connector interface providing 2 independent RF Front-End (RFFE) interfaces according to the MIPI® Alliance "System Power Management Interface Specification" and 10 General Purpose Input/Output (GPIO) pins. RFFE command execution and GPIO voltage settings can be synchronized with the sweep (sweep sequencer functionality); however, RFFE read is not supported in sweep sequencer mode.

Variants

The following variants of the RFFE-GPIO extension board are available:

- R&S ZN-B15 variant **02**, order number 1323.9355.**02**
- R&S ZN-B15 variant **03**, order number 1323.9355.**03**, including current and voltage measurements for all RFFE and GPIO pins
- R&S ZN-Z15 variant **02**, order number 1325.5905.**02**, the external version of R&S ZN-B15 variant **02**
- R&S ZN-Z15 variant **03**, order number 1325.5905.**03**, the external version of R&S ZN-B15 variant **03**

Information about the voltage and current levels is available in Section 13.3.5, "RFFE GPIO interface", on page 1715.

- For 2nd generation R&S ZNB (order nos. 1334.3330.xx) and R&S ZNBT, only the external versions R&S ZN-**Z**15 can be used.
- Currently, the external versions have to be connected via Direct Control, which in turn requires Device Control option R&S ZNB-B12.
- No matter if connected internally or externally, there is no need to perform any kind of calibration or alignment of the extension board. The analyzer firmware automatically detects and supports it. The controlled devices can be hot-plugged.
- The analyzer firmware only supports a single RFFE GPIO interface board.

Mounting the internal unit (1st generation R&S ZNB only)

The internal version R&S ZN-B15 of the RFFE-GPIO extension board uses the same slot as the bias tees (see Section 4.7.14, "Bias tees", on page 258). If bias tees are installed, you have to remove them first.

NOTICE

Risk of board and instrument damage

Please **turn off the R&S ZNB** before mounting an internal RFFE-GPIO extension board R&S ZN-B15. A hot plug installation is not supported and can damage board and instrument.

Current and voltage measurements (variants 03 only)

For test purposes, variants 03 of R&S ZN-B15 and R&S ZN-Z15 can apply constant, pin-specific output voltages not only to the GPIO pins but also to the individual RFFE pins. Furthermore it can measure the (resulting) voltages/currents at these pins using suitable shunt resistances. The firmware takes 500 samples per millisecond (and pin) and calculates the average voltages/currents over the configured measurement time.

RFFE Cable with adapters (R&S ZN-Z25)

A 2m ribbon cable for connecting a DUT to the RFFE GPIO interface – along with a set of adapters – is shipped with each RFFE GPIO extension board. It splits the 25-pin

connector of the RFFE GPIO interface to four 10-pin socket connectors. The adapters provide several different pin configurations.

This interface cable/adaptor kit can also be ordered separately from Rohde & Schwarz (R&S ZN-Z25, order no. 1334.3424.02). For a detailed pin description, see the R&S ZN-Z25 data sheet.

4.7.21 Additional removable system drive

Option R&S ZNB-B19

Option R&S ZNB-B19 provides an additional removable system drive for the R&S ZNB, including operating system and VNA firmware.

See the data sheet for the available variants of this option.

4.7.22 Extended power range

Various "B2n" Options (see tables below)

The "Extended Power Range" options reduce the minimum source power for the R&S ZNB:

Analyzer	Option	Reduced minimum source power
R&S ZNB4, 2 ports	R&S ZNB4-B22	-85 dBm
R&S ZNB4, 4 ports	R&S ZNB4-B24	
R&S ZNB8, 2 ports	R&S ZNB8-B22	
R&S ZNB8, 4 ports	R&S ZNB8-B24	
R&S ZNB20, 2 ports	R&S ZNB20-B22	-60 dBm
R&S ZNB20, 4 ports	R&S ZNB20-B24	
R&S ZNB26, 2 ports	R&S ZNB26-B22	
R&S ZNB26, 4 ports	R&S ZNB26-B24	
R&S ZNB40, 2 ports	R&S ZNB40-B22	
R&S ZNB40, 4 ports	R&S ZNB40-B24	
R&S ZNB43, 2 ports	R&S ZNB43-B22	
R&S ZNB43, 4 ports	R&S ZNB43-B24	

With extended power range, you have to configure AGC for the related a-waves manually (see Section 5.12.3.2, "AGC Manual Configuration dialog", on page 701).

4.7.23 Extended dynamic range

Options R&S ZNB4|8-B52 and R&S ZNB4|8-B54

Hardware options R&S ZNB4-B52/-B54 and R&S ZNB8-B52/-B54 increase the dynamic range of R&S ZNB4 and R&S ZNB8 up to 140 dB (typically 150 dB) for full 2-port calibrated measurements. Compared to standard equipping, the noise floor of the receivers is shifted 10 dB down by reducing the attenuation in the frontends. The trace noise for transmission measurements is also improved.

Table 4-27: Extended dynamic range options for R&S ZNB

Analyzer type	2-port	4-port
R&S ZNB4	R&S ZNB4-B52	R&S ZNB4-B54
R&S ZNB8	R&S ZNB8-B52	R&S ZNB8-B54

These options cannot be retrofitted. They are only available for **new** R&S ZNB4 and R&S ZNB8.

The extended dynamic range is particularly useful for testing passive devices with high dynamic range. It also helps you to increase the measurement accuracy for devices with high attenuation, or to increase measurement speed for devices with medium attenuation, by using wider IF bandwidths. With 10 dB more dynamic range, measurement bandwidths can be increased by a factor of 10. A typical production application that currently uses a sweep with 201 points with 10 kHz bandwidth and full 2-port calibration can now use 100 kHz to become significantly faster.

- Extended dynamic range options are not available for R&S ZNB20|26|40|43.
- Extended dynamic range and Receiver step attenuators are mutually exclusive
- Extended dynamic range and Bias Tees are mutually exclusive
- To achieve the higher sensitivity, with the extended dynamic range option:
  - The 0.1 dB compression point is reduced
  - The maximum output power is reduced

  See the R&S ZNB specifications document for details

4.7.24 Receiver step attenuators

Various Options (see tables below)

Receiver step attenuators are used to adjust the received signal levels to the input level range of the analyzer. The additional attenuation can prevent receiver compression or damage, e.g. if the DUT is a power amplifier.

The attenuation values can be set between 0 dB and 30 dB (in 10 dB steps); see "Step Attenuators" on page 532. The following step attenuators are available:

Table 4-28: R&S ZNB

Analyzer type	Port 1	Port 2	Port 3	Port 4
R&S ZNB4	R&S ZNB4-B31	R&S ZNB4-B32	R&S ZNB4-B33	R&S ZNB4-B34
R&S ZNB8	R&S ZNB8-B31	R&S ZNB8-B32	R&S ZNB8-B33	R&S ZNB8-B34

- Receiver step attenuators are not available for R&S ZNB20|26|40|43.
- Receiver step attenuators for R&S ZNB4|8 can be retrofitted at Rohde & Schwarz service.

4.7.25 R&S ZNB40 reflector module with ESD limiters

Option R&S ZNB40-B73/-U73

With the new reflectometer board (order no. 1317.8015.03), the reflectometers of a R&S ZNB40 are more robust against ESD events.

If ordered with option R&S ZNB40-B73, a new R&S ZNB40 is equipped with the new reflector module. An upgrade option R&S ZNB40-U73 is also available, however the R&S ZNB40 has to be returned to Rohde&Schwarz service for installation.

- Unlike the standard R&S ZNB40 reflectometer board, the new one does not come with and does not allow for bias tees.
- Option and upgrade require a firmware version 3.40 or higher.

4.7.26 DC inputs

Option R&S ZNB-B81

This hardware option provides four rear panel connectors labeled DC INPUT 1 ... 4 which can be used for DC measurements in different voltage ranges. The DC inputs are also needed for the measurement of the Power Added Efficiency (PAE).

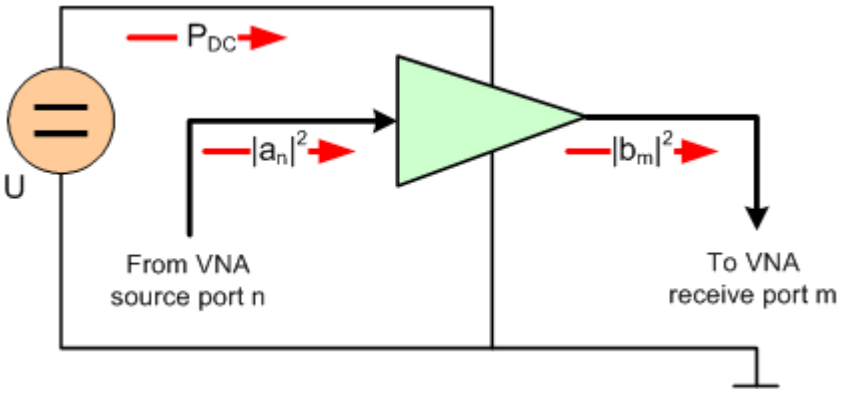
4.7.26.1 Power added efficiency

The Power Added Efficiency (PAE) is the ratio of the added RF power generated by an active two-port device (e.g. an amplifier) to the supplied DC power P_{DC}. The added RF power can be expressed as the difference between the power of the outgoing wave b_m at the output of the DUT and the power of the incident wave a_n at the input of the DUT; hence:

$$PAE = \frac{|b_m|^2 - |a_n|^2}{P_{DC}}$$

Positive PAE values indicate a gain in the RF power, negative values an attenuation. The PAE is always smaller than 1.

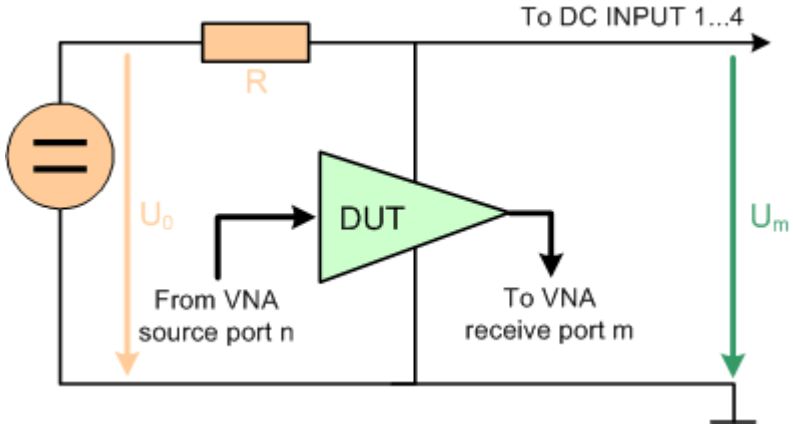
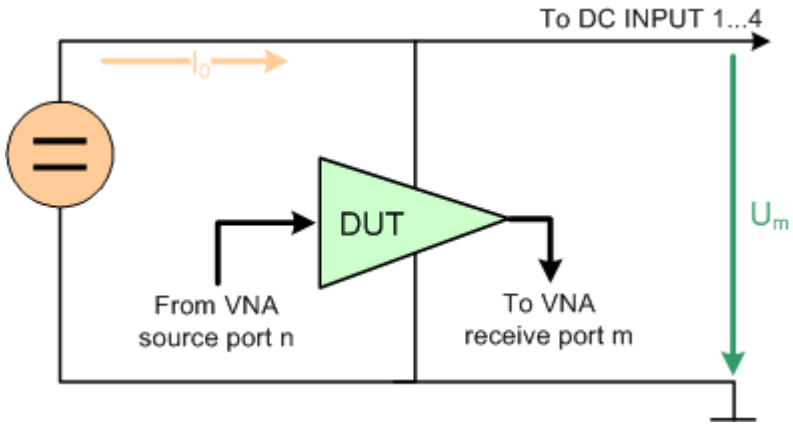
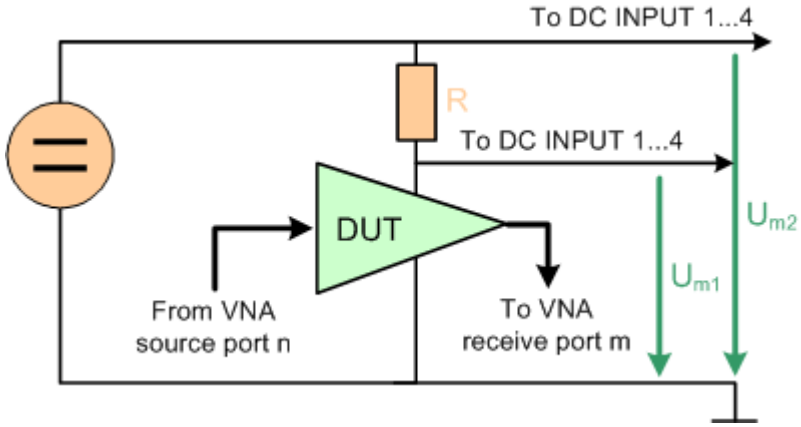
The PAE measurement is based on the standard test setup for forward S-parameter measurements on a 2-port DUT. An additional measurement to determine the supplied power P_{DC} is required.



DC power measurement

The DC power P_{DC} supplied to the DUT can be measured using one or two of the four DC inputs DC INPUT 1...4 at the rear panel. The Power Added Efficiency (PAE) dialog suggests different measurement types, involving different test setups. The measurement types depend on the properties of the DC power supply (constant current I_0 or constant power U_0) and an optional precision resistor R used to measure the DC current. Depending on the measurement type, one or two of the values I_0, U_0, and/or R must be entered as parameters of the PAE calculation.

Table 4-29: PAE measurement types

Measurement type / Circuit diagram	Input parameters / Description
Constant voltage source 	**U_0, R** The DC power supply provides a constant voltage U_0; a precision resistor R is connected in series to the DUT. A single measurement of the DC voltage at the DUT U_m provides the current and hence the DC input power: $P_{DC} = U_m * I = U_m * \frac{U_0 - U_m}{R}$
Constant current source 	**I_0** The DC power supply provides a constant current I_0. The analyzer measures the DC voltage at the DUT U_m and calculates the DC input power as: $P_{DC} = U_m * I_0$
Voltage and current 	**R** Voltage and current of the DC power supply can be both unknown; a precision resistor R is connected in series to the DUT. Two simultaneous DC voltage measurements provide the DC input power: $P_{DC} = U_{m1} * I = U_{m1} * \frac{U_{m2} - U_{m1}}{R}$

4.7.27 USB-to-IEC/IEEE adapter

R&S ZVAB-B44

Hardware option R&S ZVAB-B44 (order no. 1302.5544.03) comprises an adapter and driver software for controlling external devices via IEEE 488 / IEC 625 (GPIB). The driver software is installed on the network analyzer. Connect the USB port of the adapter to any of the master USB connectors on the front or rear panel of the analyzer. Connect the GPIB port of the adapter to the external device.

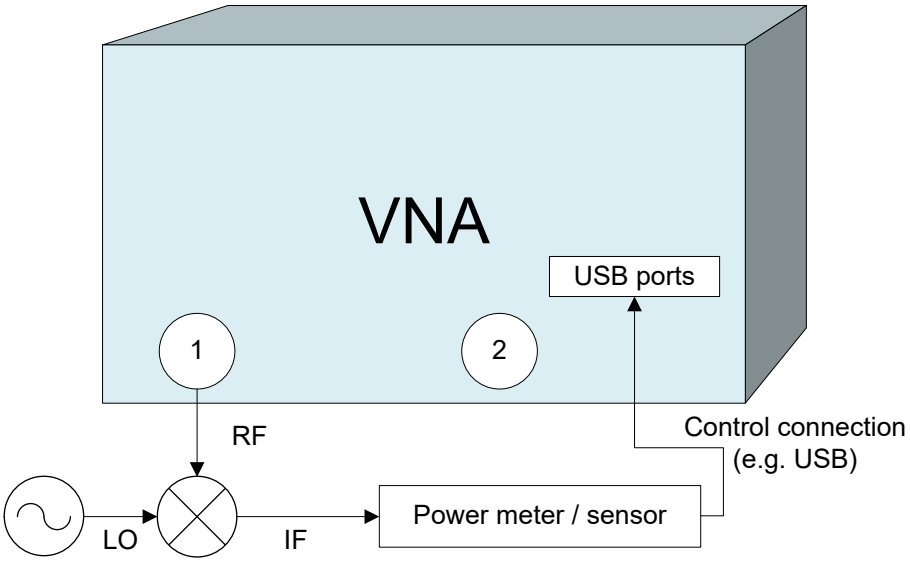
This hardware option is required, if you want to use GPIB for controlling external devices from the R&S ZNB. The current B10 option does not offer this functionality.

To get the adapter to work, you have to install the NI VISA library on the R&S ZNB.

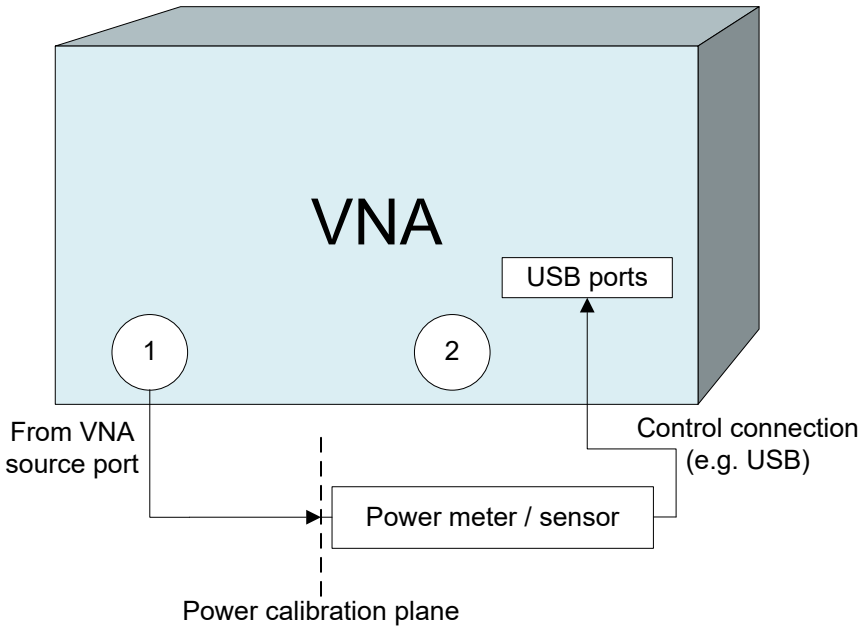
4.7.28 External power meters

The connection of an external power meter to the R&S ZNB can serve different purposes.

- Extended measurement functionality: Each external power meter represents an additional receive port. External power meters increase the number of RF input signals of a DUT that the analyzer can measure simultaneously. They can also provide accurate results for signals at inaccurate or unknown frequencies. A typical example is a mixer measurement with an unknown LO signal (and therefore unknown IF output frequency).



- Power calibration: An external power meter can measure the exact signal power at an arbitrary point in the test setup (reference plane) and thus provide the reference values for a power calibration. A typical example is a source power calibration for an arbitrary analyzer port.



External power meters must be configured with their connection type and device address before they are available as additional receivers (System – [Setup] > "External Devices" > "Power Meters"). Configured power meters appear in many control elements of the R&S ZNB, e.g. in the port configuration and in the power calibration dialogs.

4.7.28.1 Zeroing

Zeroing calibrates the external power meter by adjusting its reading at zero signal power. For this purpose, the RF cable between the analyzer and the power sensor must be disconnected (see tips below!). R&S power sensors and power meters automatically detect the presence of any significant input power. This aborts zeroing and generates an error message. Zeroing can take a few seconds, depending on the power meter model; refer to the documentation of your external power meter for more information.

Repeat zeroing

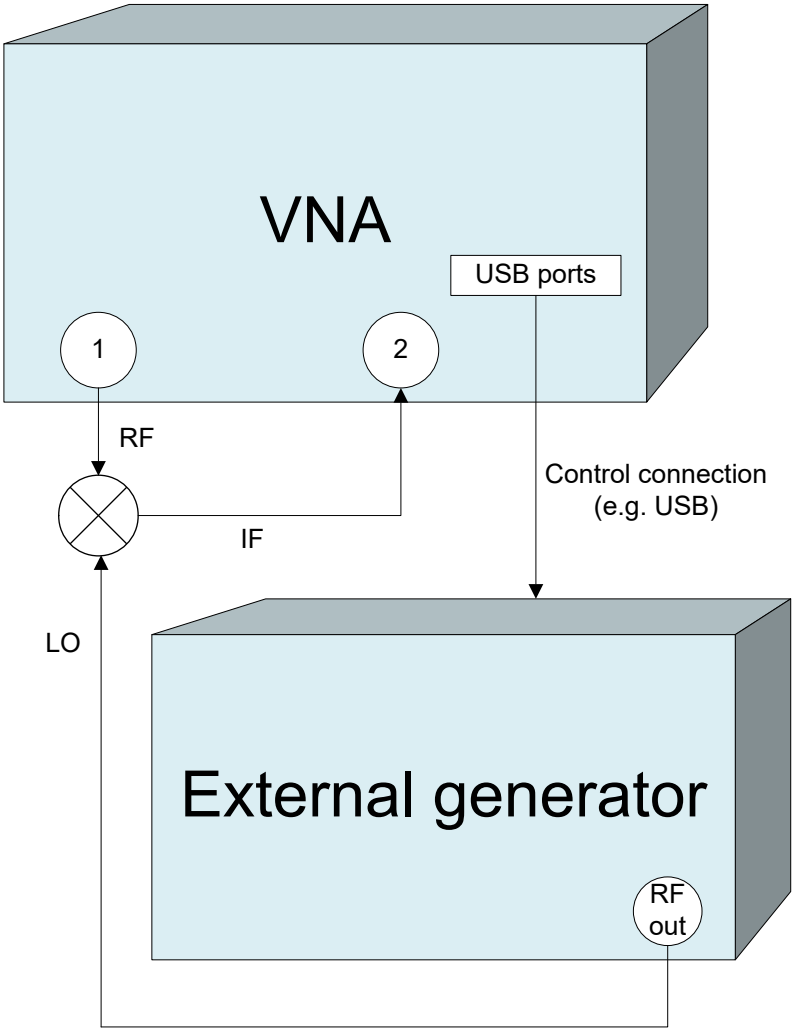
- During warm-up after switching on or connecting the instrument
- After a substantial change of the ambient temperature
- After fastening the power meter to an RF connector at high temperature
- After several hours of operation
- When low-power signals are to be measured, e.g. less than 10 dB above the lower measurement limit.

A reset of the network analyzer does not affect the last zeroing result.

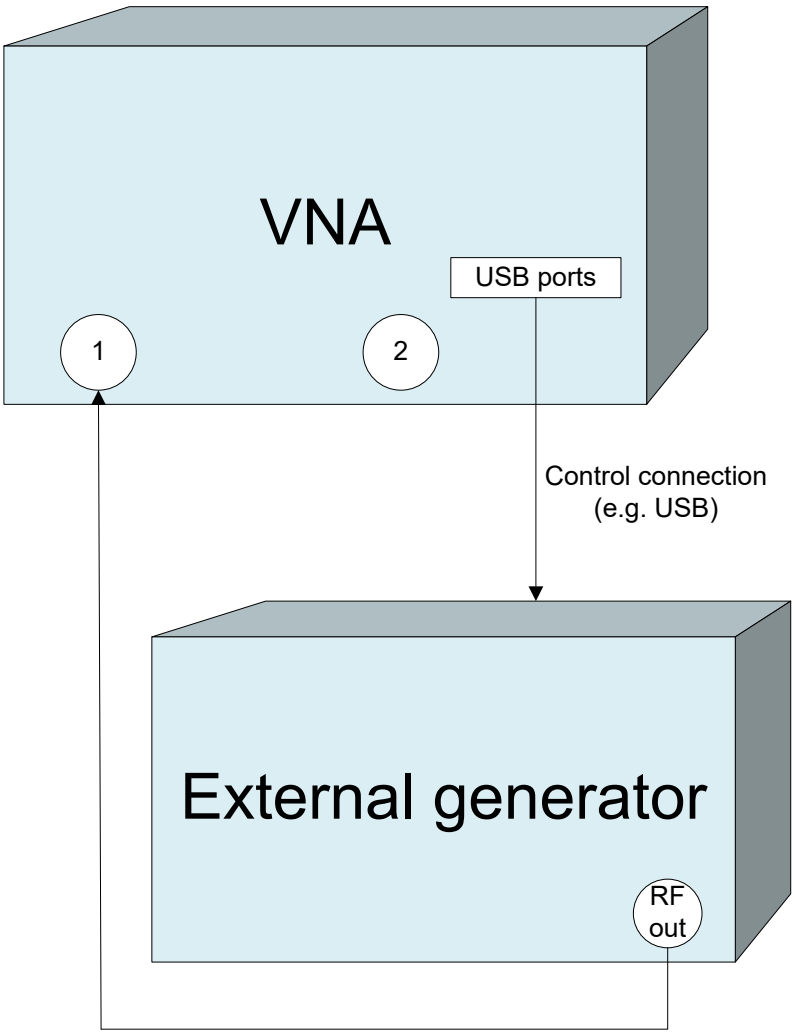
4.7.29 External generators

The connection of an external generator to the R&S ZNB can serve different purposes.

- Extended measurement functionality: Each external generator represents an additional source port. External generators increase the number of RF input signals for the DUT. A typical example is a mixer measurement with a 2-port analyzer, where an external generator provides the LO input signal.



- Power calibration: An external generator can provide the reference signal for a source or receiver calibration. A typical example is a receiver power calibration using a measured wave b_1.



External generators must be configured in the System – [Setup] softtool before they are available as additional sources. Configured generators appear in many control elements of the R&S ZNB, e.g. in the "More Ratios", "More Wave Quantities", port configuration and power calibration dialogs.

Reference frequency

To ensure frequency accuracy and frequency stability in a test setup where different devices represent the signal sources and receivers, it is advisable to use a common reference frequency. Accurate frequencies are particularly important if external generators are used for measurements with narrow measurement bandwidths.

A common reference frequency can be established in different ways:

- Use the analyzer as master device: Set the analyzer to "Internal" frequency reference (System – [Setup] > "Freq. Ref." > "Internal") and operate all other devices in external reference mode using the reference clock signal from the REF OUT connector on the analyzer's rear panel.

- Use another device as master: Set the analyzer to "External" frequency reference and synchronize it (and all other devices) to the master's reference clock signal, fed in at the REF IN connector on the analyzer's rear panel.

Fast sweep mode and conditions

In list mode the external generator steps through a predefined list of frequencies or signal powers. This mode can be used to accelerate the measurements involving external generators. If "Fast Sweep" is activated in the "External Generators" dialog, the analyzer compiles a list of the stimulus values (frequencies and powers) in all channels and transfers it to the generator. The list is automatically updated and retransferred whenever the channel settings are changed.

The analyzer uses a trigger handshake mechanism in order to control the generator's list mode:

- The generator sends an EXT GEN BLANK signal to pin no. 22 of the USER PORT connector on the rear panel of the analyzer to show that it is ready to step to the next frequency or power value in the list.
- The analyzer transmits an EXT GEN TRG signal at pin no. 21 of the USER PORT connector in order to switch the generator to the next point in the list. Afterwards the analyzer waits for the next EXT GEN BLANK signal.

If the USER PORT connection is interrupted during the measurement, the sweep is halted. It is continued after a "Restart Sweep".

If the number of sweep points exceeds the maximum number of entries in the list (depending on the generator type), the analyzer must interrupt the sweep in order to send a new list and complete the stimulus information. This generally slows down the measurement.

Measurement process for external generators

The measurement process for **external generators** "Gen 1", "Gen2" ... differs from the measurement process for internal source ports:

- An external generator always represents a permanent signal source that is switched on for all partial measurements.
- The external source is measured in the first partial measurement where an internal source is active. This means that no separate partial measurement for the external generator signal is needed. If no internal source is needed at all, the external source is measured in the first partial measurement.

4.7.30 External switch matrices

4.7.30.1 Overview

From the perspective of the VNA, the purpose of a switch matrix is to extend the number of test ports. I.e. it converts an N-port network analyzer into an N'-port network analyzer (N'>N) without modifying the instrument itself. The increased number of test

ports can reduce or even eliminate the manual reconnections of the DUT, resulting in a higher measurement speed, reliability and repeatability.

The firmware of the R&S ZNB is able to control switch matrices from Rohde & Schwarz directly. For example, a 4-port R&S ZNB26 in combination with two R&S ZN-Z86 can work as a multi-port network analyzer with up to 48 ports, seamlessly performing the required matrix switching operations.



left = 4-port R&S ZNB26/40/43 with R&S ZN-ZA86 and semi-rigid cable set R&S ZN-ZB26 var. 02
middle = 4-port R&S ZNB26/40/43 with R&S ZN-Z86X and semi-rigid cable set R&S ZN-ZB26 var. 03
right = 4-port R&S ZNB26/40/43 with 2 x R&S ZN-Z86 and semi-rigid cable set R&S ZN-ZB26 var. 05

See Section 4.7.30.6, "Cable sets", on page 282.

While it is allowed to connect multiple matrices to a R&S ZNB, matrix cascading is **not** supported.

An N-port analyzer is always limited to measuring N signals simultaneously. I.e. even though the DUT is fully connected to the switch matrix, you cannot measure all its b-waves simultaneously. For example, consider a DUT with 6 unbalanced ports, connected to a 2-port analyzer via a 2x6 matrix (e.g. a R&S ZN-Z8x base unit without extensions). For each stimulus port, 5 sweeps are required to measure the resulting b-waves, 6·5 = 30 sweeps in total.

Furthermore, each additional sweep requires at least one matrix switching procedure. This procedure involves command processing and physical switching and hence can take some time to complete. For mechanical switches (such as R&S ZV-Z81 variant 66), in particular, switching is slow and causes mechanical wear. To optimize the measurement setup w.r.t. speed and resources, the characteristics of the available switch matrices have to be accounted for.

4.7.30.2 Matrix setup and operation

Setting up a switch matrix at the R&S ZNB firmware typically involves the following steps:

1. Establish the physical connection via the appropriate management interface (USB or LAN or Direct Control).
2. Register the matrix (as managed object).

3. Define the RF configuration:
   a) Configure the matrix-VNA-connections according to the existing (or planned) physical connections between VNA test ports and matrix VNA ports.
   b) Assign the matrix test ports and the remaining VNA ports (i.e. the VNA ports not connected to a matrix VNA port) to DUT test ports.

After this initial setup, the R&S ZNB takes control of the attached matrices: it allows you to configure the test ports and dynamically establishes the required matrix routes according to the current measurement task.



To make a measurement, set up a channel where all unused logical ports are disabled.

Switch matrices are global resources: their configuration is not part of a recall set. However, a recall set contains information about the required matrices and their RF connections. At the time the recall set is loaded, the R&S ZNB checks whether the same matrix setup is still active. If not, a wizard guides the user through the matrix configuration.

4.7.30.3 RF connections and matrix connectivity

Depending on the design of the switch matrix and the RF connections between VNA and matrix, the following limitations can occur:

- Certain matrix test ports are not available for measurements
- Certain transmission measurements are not possible

Example:

The 4x24 extension of switch matrix R&S ZN-Z84 consists of 2 separate 2x12 submatrices. Matrix VNA ports of the "left" submatrix cannot be connected to the test ports of the "right" submatrix and vice versa. If none of the two VNA ports of a submatrix is connected to the VNA, then obviously you cannot use the test ports of this submatrix for measurements.

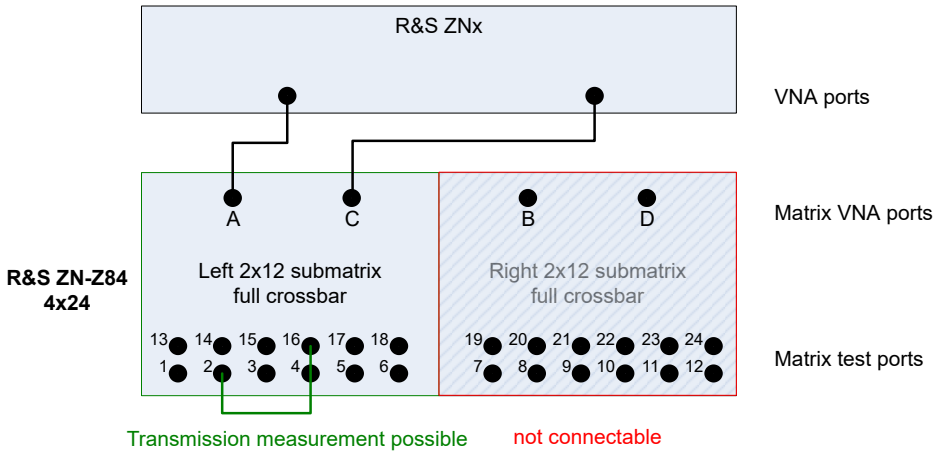


Figure 4-35: Unconnected right submatrix

With the same extension type, connecting one of two submatrix VNA ports enables reflection measurements for the corresponding matrix test ports. However, as long as a second connection between the VNA and this submatrix is missing, "intra-submatrix" transmission measurements are still impossible.

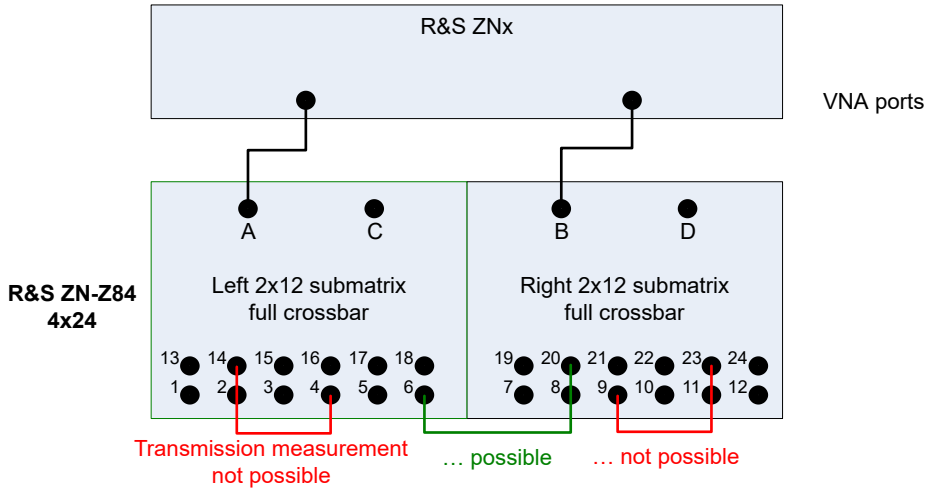


Figure 4-36: Partially connected submatrices

Basically the same argument holds true for variant 16 of switch matrix ZV-Z82, which consists of 4 separate 1x4 submatrices and that allows "inter-" but no "intra-submatrix" transmission measurements.

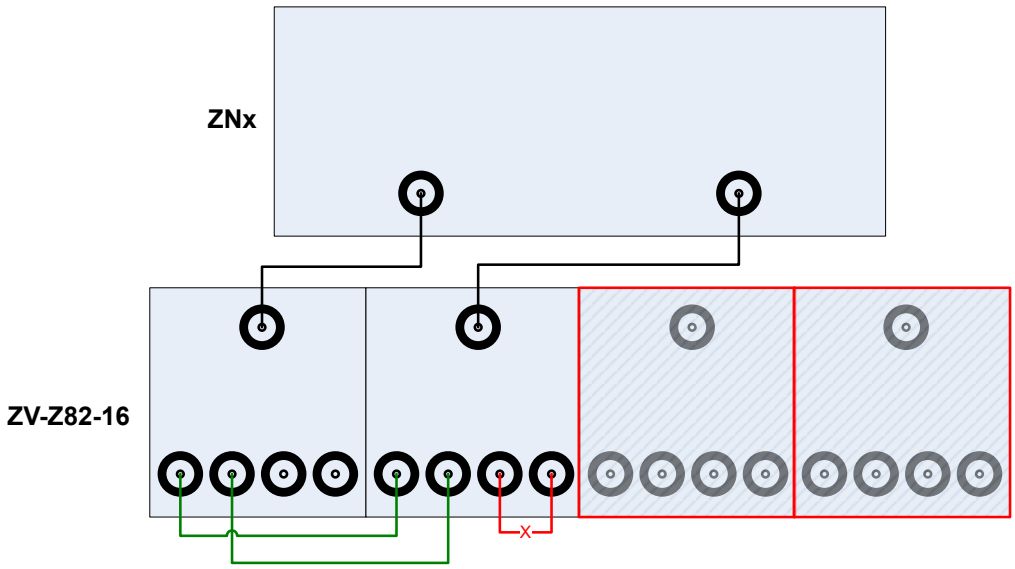


Figure 4-37: ZV-Z82-16: limited connectivity

4.7.30.4 Multiple paths: precision vs. speed

A switch matrix can offer multiple routes to a given matrix test port. Hence, measurements can be performed using different physical paths, where a *path* consists of the traversed VNA connections and matrix routes.

These paths can have different characteristics - in particular if the corresponding matrix routes differ in the number of semiconductor or mechanical switching functions that have to be traversed.

Example:

For switch matrix ZV-Z82-09, every test port can be reached via two different routes, traversing either one or two semiconductor switches:

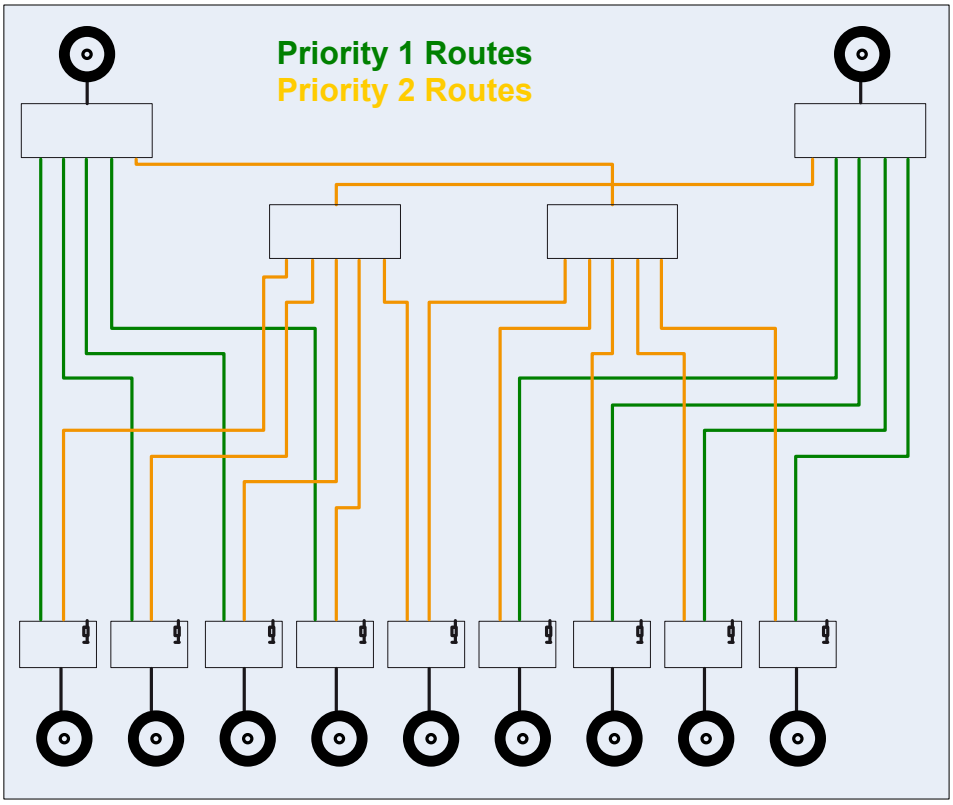


Figure 4-38: ZV-Z81 routes and priorities

Example:

For an R&S ZN-Z8x, each route traverses exactly one of the equipped 2x6 modules. The overall route quality is determined by the number of electronic solid state switches traversed on this 2x6 module:

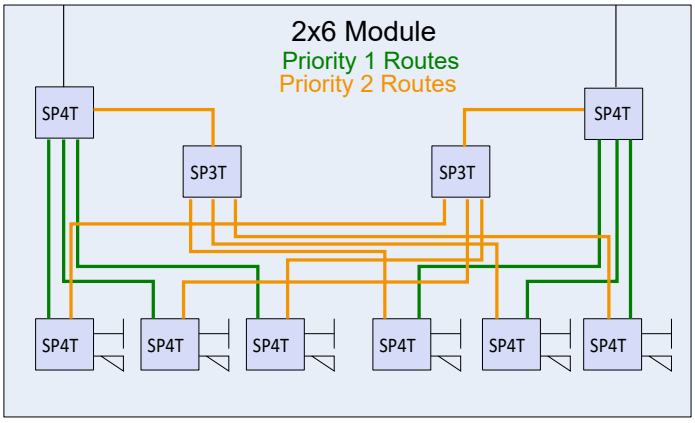


Figure 4-39: Routes and priorities

See the specifications document of the respective switch matrix for details on the available routes.

For every supported switch matrix, the available routes are prioritized according to the number of switches they traverse (the rectangles in Figure 4-38).

- To obtain the highest measurement *precision*, the driving port uses the "best possible" (highest priority) route. Because this logic can lead to additional matrix switching procedures, it can result in a reduced measurement speed.
- If the focus is on measurement *speed*, the number of matrix switching procedures is minimized, disregarding a possible loss in measurement precision.

As for "slow" management connections (via USB or LAN) the fundamental trade-off between precision and speed cannot be resolved, the R&S ZNB allows you to select the optimization to be performed.



Always choose the *precision* optimization if the management connection is established using Direct Control (see Section 4.7.18, "Device Control", on page 259). The switching speed of the matrix in this case is in the same range as the source switch of the VNA.

4.7.30.5 Multiport calibration

With switch matrices, a signal loss of up to 25 dB is possible, so (full) calibration is a must. For a high number of test ports, manual calibration is time-consuming and error-prone, so automatic calibration – using a multiport calibration unit – is the obvious way to go.



A multiport calibration unit also contains switched paths, imposing additional losses during the calibration sweeps and reducing the accuracy of certain calibration measurements. The reduced calibration accuracy, in turn, can result in a significantly reduced measurement accuracy for certain DUTs and measurements.

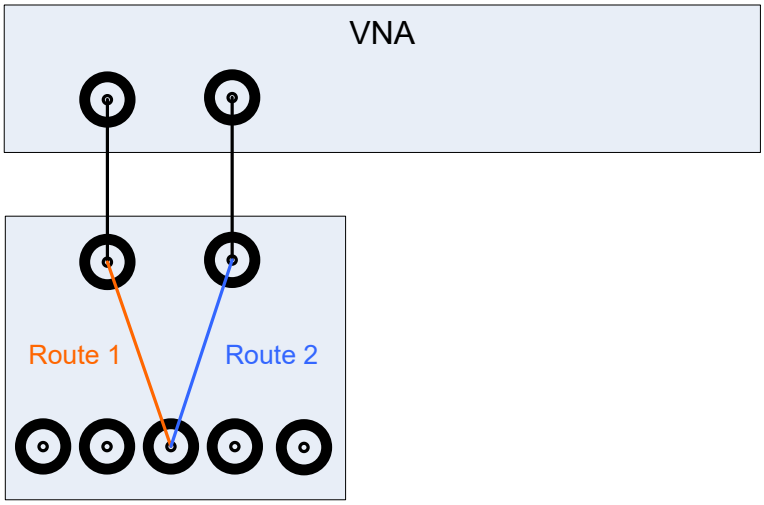
For this reason, we recommend to:

- Adjust the power and bandwidth settings during calibration.
  See "Recommended channel settings during calibration" on page 280.
- Use the "Optimized Port Assignment" proposed by the analyzer firmware.
  See "Optimized port assignment" on page 280.

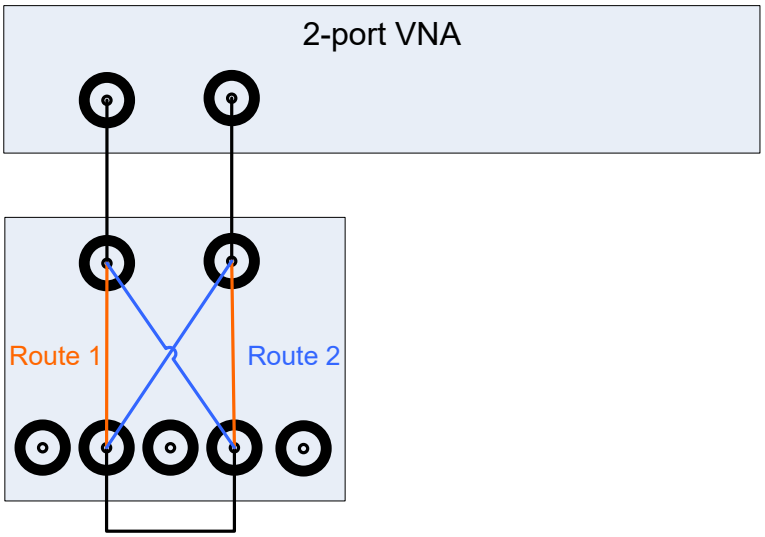
Multiple matrix paths and calibration

During calibration, correction data are determined for all possible paths from the VNA to the related test ports. For the currently supported matrices, a maximum of 2 paths per test port are available, resulting in a maximum of:

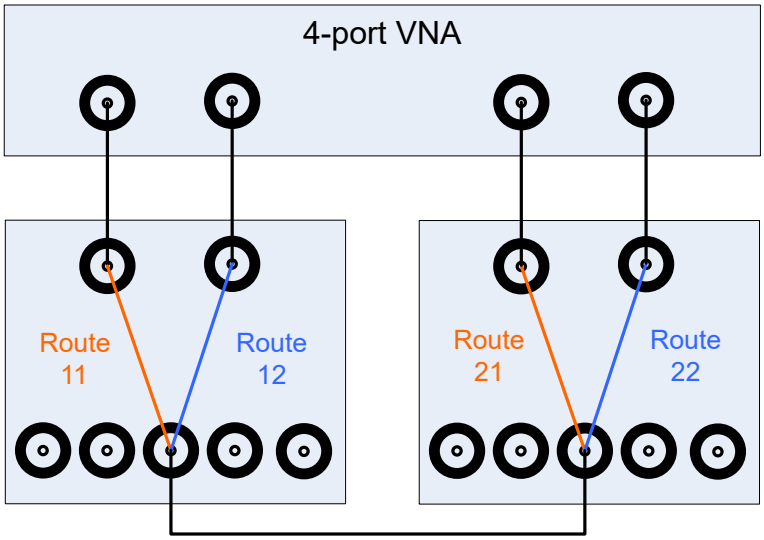
- 2 paths per reflection measurement



- 2 paths per transmission measurement for a 2-port VNA:



- 4 paths per transmission measurement for an N-port VNA with N≥4



This kind of "multipath calibration" offers the following additional benefit:

During manual calibration, the measured reflection/transmission coefficients are presented as memory traces – one per path. By comparing these traces, it can be possible to track down hardware problems (cables, connectors, matrix, ...) already during calibration.

Passive load matches

For matrix setups, calibration involves measuring the **passive load match** of all involved matrix test ports, i.e. the reflection behavior of those matrix test ports that are currently **not** switched to a (matrix) VNA port.

To measure these reflections, the source signal is routed through the matrix and the calibration unit – just like for (unknown) Through measurements. In this case, however, the signal is **terminated** at the receiving matrix test port by switching this matrix test port to **passive load match** state.

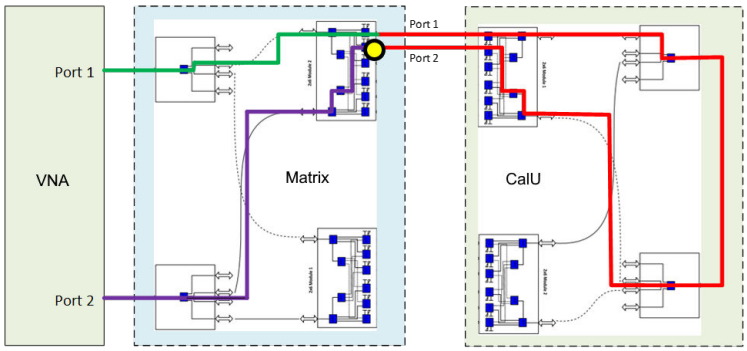


Figure 4-40: Through vs. passive load match measurement (matrix R&S ZN-Z8x, calu R&S Z-154)

The (comparatively small) reflected signal once again passes the calibration unit and matrix before it is received at the VNA (port 1, in the figure above). The additional los-

ses imposed by the backward path through the calibration unit can result in poor signal to noise ratios.

Recommended channel settings during calibration

The straightforward countermeasure against poor signal to noise ratios during calibration is to increase the stimulus power and to reduce the measurement (IF) bandwidth. To do so:

1. Disable Auto Power Reduction for Cal Unit
2. Set the stimulus power to 0 dBm during calibration
3. Set the IF bandwidth to 1 kHz during calibration.
   (Reduces S/N, but increases measurement time)



- Revert the channel power and bandwidth to their intended values after calibration.
- "Auto Power Reduction for Cal Unit" is a global, persistent setting. Consider enabling it for other measurements.

Optimized port assignment

To mitigate the problem of excessive path losses, the auto-cal logic selects a path with the lowest possible attenuation when measuring a matrix test port's passive load match. The result, however, depends on:

- The structure of the connected matrix or matrices
  (See Section 4.7.30.4, "Multiple paths: precision vs. speed", on page 275)
- The structure of the calibration unit
  (See Figure 4-40)
- How the test ports are connected to the calibration unit ports

For best results, the VNA firmware calculates an optimized assignment between test ports and calibration unit ports, given the structure of the matrices and calibration unit in use.

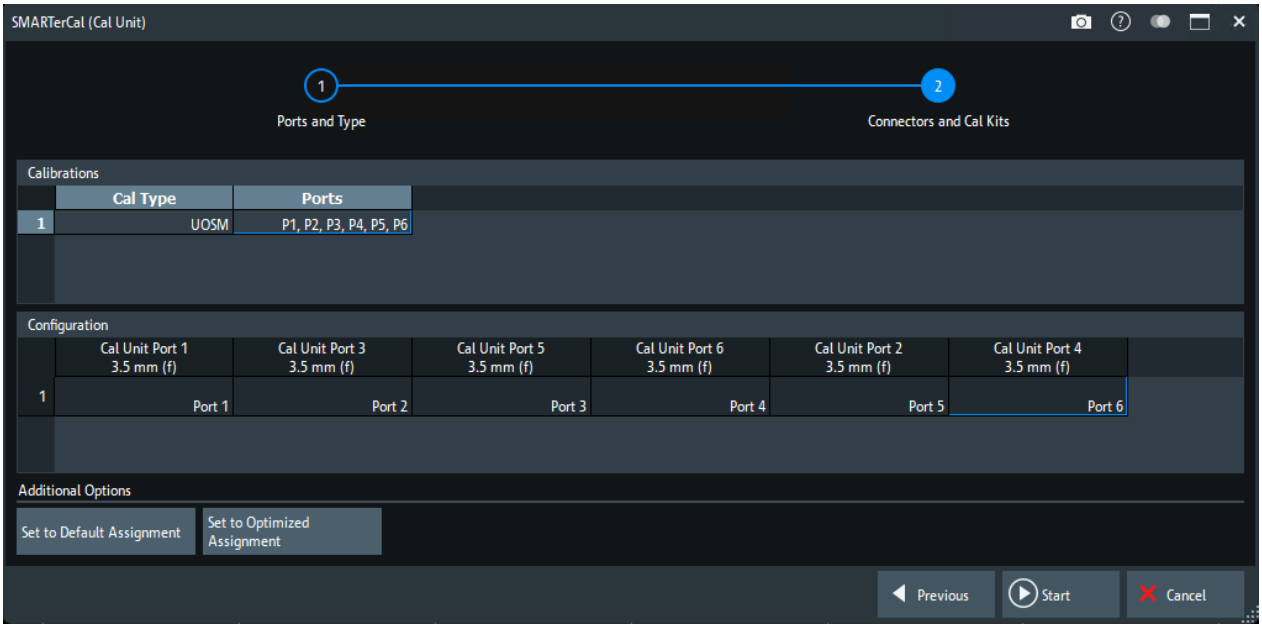


Figure 4-41: Optimized port assignment for 6-port R&S ZN-Z154

If your measurement setup contains VNA ports that are not connected to a switch matrix, it is recommended to include (at least) one of them in the calibration. In the optimized port assignment, it is used as the source port for all passive load match measurements, and as the common port for all (unknown) Through measurements during calibration.

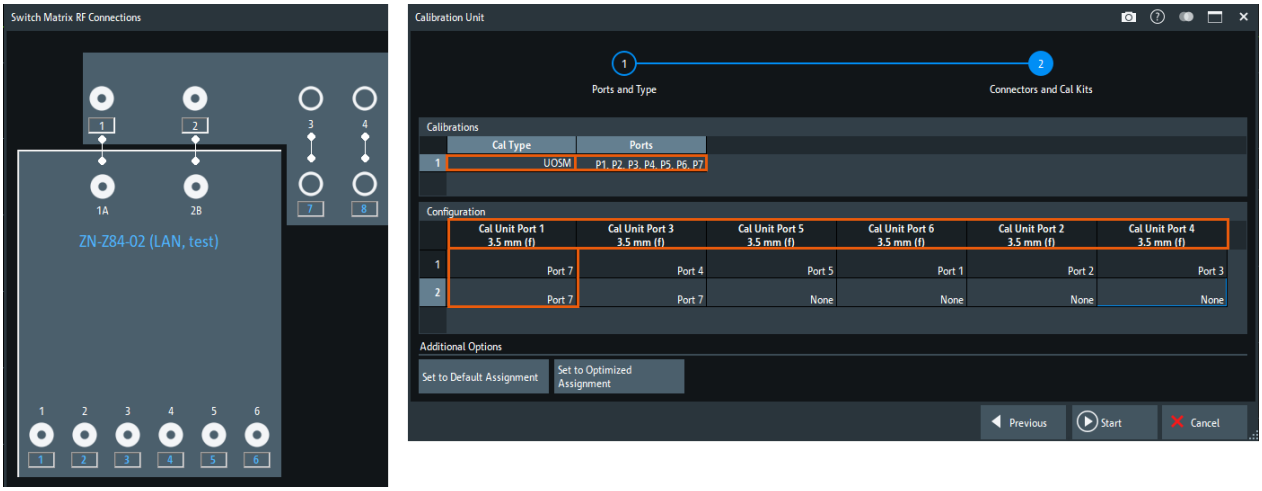


Figure 4-42: Direct VNA ports

If, even with "Recommended channel settings during calibration" on page 280 and optimized port assignment, you are not confident with the measurement results:

- Try out a calibration unit with fewer ports (fewer switches)
  See R&S ZN-Z5x, R&S ZN-Z15x, R&S ZV-Z5x
- Go for manual calibration ...

4.7.30.6 Cable sets

For some combinations of analyzers and switch matrices, dedicated RF connection cable sets are available for benchtop operation.

For example, cable set R&S ZN-Z28, N (m) to 3.5 mm (m), allows you to connect a 4-port R&S ZNB4|8 to two R&S ZN-Z84 with 2 matrix VNA ports.

Table 4-30: Semi-rigid RF cable sets R&S ZN-ZB26

Order no.	Cable type	VNA	Switch matrix
1328.8911.02	2.92 mm (f) to 2.92 mm (m)	R&S ZNB26|40|43	R&S ZN-Z86 2 or 4 matrix VNA ports
1328.8911.03			R&S ZN-Z86X
1328.8911.05			R&S ZN-Z86 2 units with 2 matrix VNA ports

4.7.31 External DLLs

The R&S ZNB firmware provides a plug-in interface that allows you to extend the VNA firmware with custom functionality (measurements, device control, ...) .

The plug-in interface allows the external software to interact with the firmware before and after a change of the:

- channel
- drive port
- sweep segment
- sweep point

The plug-in then can perform arbitrary actions and eventually return data to the VNA firmware:

- as custom traces
- as input values for math functions for the calculation of other traces

To calculate a custom trace, the plug-in can make use of all data access points in the channel calculation chain. See Section 4.1.5.1, "Channel data flow", on page 89).

Furthermore, it can communicate with other processes / peripherals that are not part of the VNA firmware or host OS.

4.7.31.1 Basic use case

A basic use case is the integration of a power supply into an arbitrary VNA sweep.

Suppose that you want to test an amplifier with various bias settings. To do so, you can perform a CW sweep to measure S-parameters with different gate voltages applied to the amplifier at each sweep point.

With one power supply channel connected to the amplifier's gate, and another channel connected to the amplifier's drain, the measurement could proceed as follows:

1. At the beginning of the sweep, the power supply is initialized (e.g. voltage setting and current limit for each power supply channel) and then switched ON.
2. At the beginning of each sweep point, the plugin sets the power supply gate voltage to the next value.
3. At the end of each sweep point, the plugin reports the power supply reading of the gate voltage and drain current to the VNA. These readings are displayed in dedicated traces.
4. At the end of the sweep, the power supply is switched OFF.

4.7.31.2 Technology

The plug-in interface is a traditional C style DLL interface. Suitable DLLs can be compiled using different platforms, such as C or MATLAB. A plug-in is installed in the firmware as a `*.zip` file, containing the external DLL and, possibly, additional files to be kept with the DLL.

An SDK for external DLLs is provided with the analyzer firmware on the Rohde& Schwarz internet pages.



For details, see the HTML help topics that are distributed with the plugin SDK.

Interactions of the external DLL with analyzer firmware and external devices are synchronized with the measurement. Synchronization is done via callbacks from the FW, at so-called interrupts. At an interrupt, the measurement takes a break and waits for the FW to continue. During an interrupt, the plugin can access external devices and return results to the FW.

The VNA firmware can abort the processing in the external DLL at any time, e.g. on a [Preset].

The DLL can handle certain trace types that must be uniquely named within the DLL. A naming scheme for traces allows the firmware (and hence the user) to request a particular trace.

If an external DLL is configured for permanent use, it performs a particular task (such as switching the state of a DUT or controlling external devices) without providing data.

4.7.31.3 Trace calculation in the channel calculation chain

To enable the calculation of trace results using data from external DLLs, a dedicated calculation step for external DLLs is inserted directly before the averaging step in the calculation chain (see Section 4.1.5, "Data flow", on page 89).

This position allows maximum flexibility, because:

- The trace calculation chain is limited to one input for one output
- The position allows the plugin to use the results of the averaging step, and the trace data is calculated in the same way as for other calculation steps
- All internal measurement types can be used as input for the new calculation step

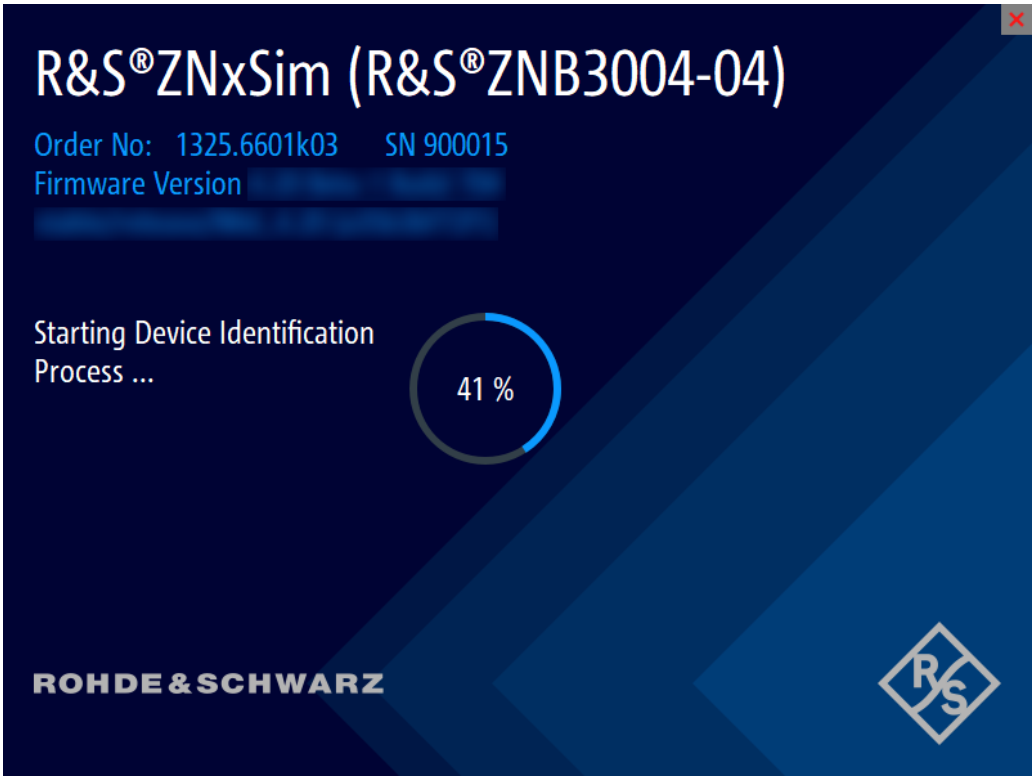
The new calculation step is only enabled if a plugin is installed (and active), and a custom trace is requested.

The new calculation step serves all active external DLLs. When custom traces are requested, the firmware asks the related DLLs which input traces they need to calculate "their" traces. During the calculation, the requested inputs are provided to the DLL, which then calculates its requested traces.

Due to its position, all internal trace types are allowed as inputs. External types are also allowed if no circular dependency is created between them.

4.7.32 R&S ZNXSIM

With the R&S ZNXSIM PC simulation, you can have all the functions of an R&S ZNA, R&S ZNB, R&S ZNBT or R&S ZND on your desktop. Always accessible, even when you do not have access to the hardware.



The same software that runs on the instrument (firmware) is used to run the simulation.

- The R&S ZNA firmware simulates all available R&S ZNA models. It can be installed in parallel with an R&S ZNB or R&S ZND firmware, but only one of them can be run at a time.
- The R&S ZNB3000 firmware simulates all R&S ZNB3000, R&S ZNB, R&S ZNBT, and R&S ZND models. It also comprises the documentation for all these instruments.

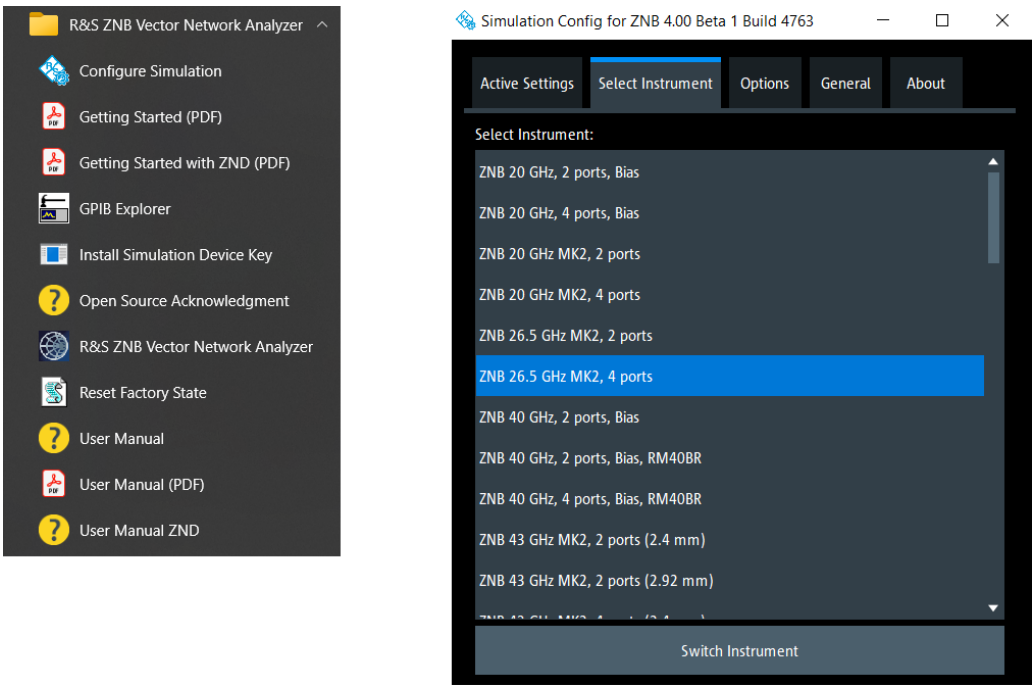


Figure 4-43: Instrument selection

For information on installation and licensing, see the R&S ZNXSIM Getting Started manual available at https://www.rohde-schwarz.com/manual/ZNXSIM.

4.7.32.1 Simulation data

The firmware simulation displays traces. To this end, it loads simulation data from one or more files, by default from `C:\Program Files\Rohde-Schwarz\Vector Network Analyzer\ZNB\Resources\Simulation\Measurements\InputBufferGroupSimulation.s4p`.

This Touchstone file contains the 16 S-parameter traces of a 4-port network (4x4 S-matrix), measured at 1193 points in the frequency range from 0 GHz to 70 GHz. The S-Matrix "simulates" a linear network (DUT) connected to VNA ports 1 to 4.

For 2-port VNAs, the left upper 2x2 submatrix is used.

For instruments with N ≥ 4 ports (on the VNA and connected switch matrices), the firmware replicates the loaded S-matrix $\lceil N/4 \rceil^2$ times, and uses the left upper NxN matrix:

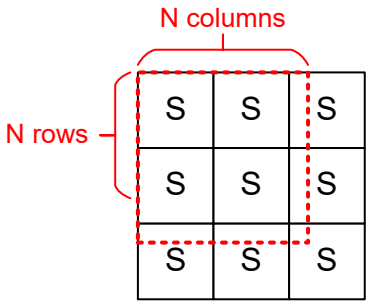


Figure 4-44: S-matrix replication

By default, the firmware simulation applies `InputBufferGroupSimulation.s4p` to all channels in all setups.

For each setup in the firmware simulation, you have three possibilities to define and refine the simulated DUT.

1. Load setup-specific simulation data from file.
   For the related setup, they take precedence over the default simulation data.

   If the number of physical test ports is larger than the "rank" of the S matrix the firmware loads from the specified file (N for `sNp` files), the previously mentioned replication logic applies.

2. Load channel-specific simulation data from file.
   For the related channel, they take precedence over the data of the setup-specific and default simulation data.
   Again, the previously mentioned replication logic applies.

3. Load port- and channel-specific S-parameter data from file.
   For the related ports, they take precedence over channel-specific, setup-specific, and default simulation data.

To determine S_{ij} at a channel frequency, the firmware interpolates between the two "closest" frequencies of the S-parameter trace ultimately loaded for S_{ij}.

4.7.32.2 Simulation noise

The firmware simulation allows you to add noise to the simulated data, i.e. a random small value added to the simulation data loaded from file. This random value changes every sweep and every sweep point in time, with the effect of seeing more "realistic" fluctuating traces, in particular in continuous sweep mode.

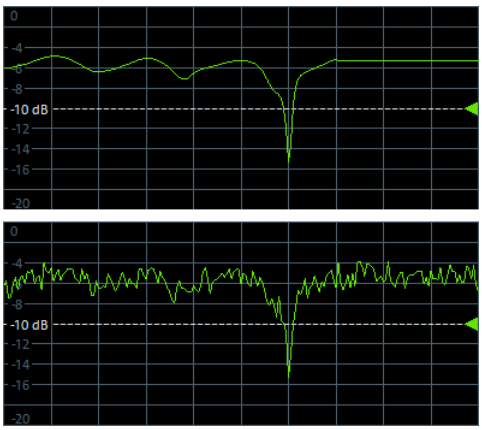


Figure 4-45: Simulation noise

Simulation noise can also be activated or deactivated from the analyzer GUI.

- Simulation noise can also be activated or deactivated from the analyzer GUI. See Section 5.14.6, "More tab", on page 787.
- Make sure to deactivate "Simulation Noise" before exporting traces to file (see Section 4.4.2, "Trace files", on page 143). Otherwise the exported results comprise an "artificial" random component.

5 GUI reference

This section describes the Graphical User Interface (GUI) of the analyzer.

The most convenient way to access the GUI functions is via Softtools. Hence the GUI reference is structured accordingly.

The softtools, in turn, can be opened via the keys on the front panel of the analyzer, via the on-screen Hardkey panel or via the items in the main menu. For details, see Section 5.1, "Function keys and softtools", on page 289.

In case a GUI function can also be performed via remote control, one or more links at the end of the function description point to the related remote control commands.



For a general overview of the analyzer's capabilities and their use, refer to Section 4, "Concepts and features", on page 79.

5.1 Function keys and softtools

Most of the Function keys serve as "openers" for an associated softtool in the analyzer GUI.

By default, the following "opener logic" is applied:

- If the associated softtool is not displayed, pressing the hardkey
  - opens the associated softtool
  - activates its first enabled tab (default) or the last used tab (see "Use Default Tab for Hardkey" on page 816)
  - activates the first enabled input control on this tab (if any)
- If the associated softtool is already displayed, pressing the hardkey
  - activates the next enabled tab on the associated softtool (cyclically)
  - activates the first enabled input control on this tab (if any)

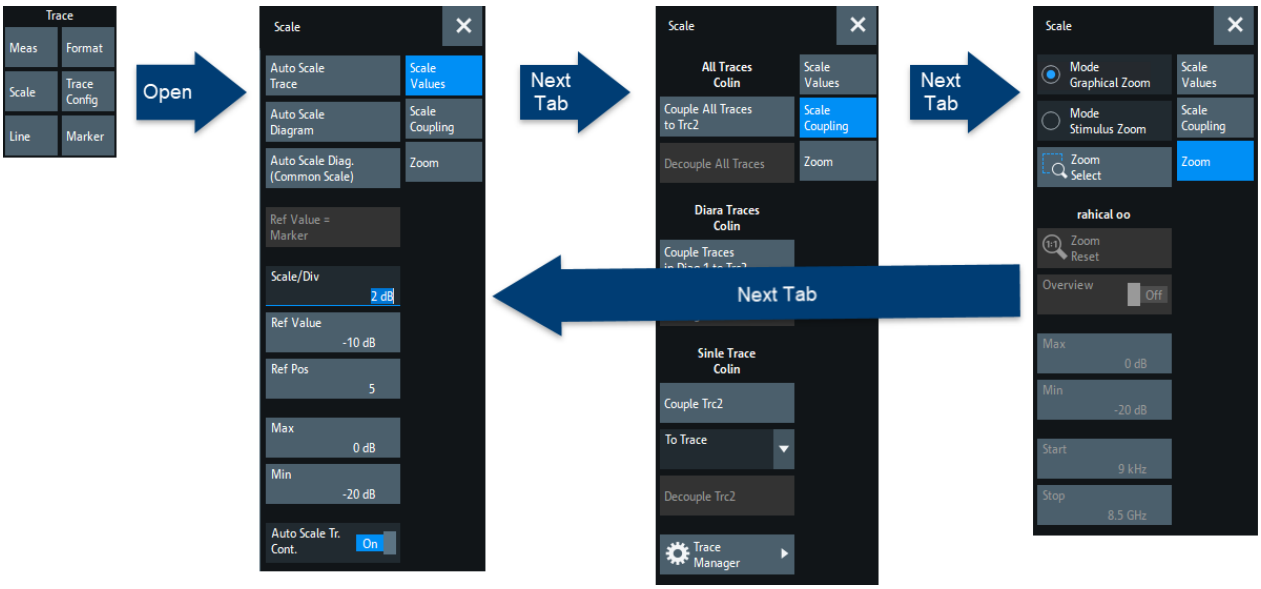


Table 5-1: Function keys and softtools

(Virtual) Hardkey	Keyboard Shortcut	Related Softtool	Action
Trace – [Meas]	Alt + Shift + A	Meas softtool	default
Trace – [Format]	Alt + Shift + B	Format softtool	default
Trace – [Scale]	Alt + Shift + C	Scale softtool	default
Trace – [Trace Config]	Alt + Shift + D	Trace Config softtool	default
Trace – [Line]	Alt + Shift + E	Lines softtool	default
Trace – [Marker]	Alt + Shift + G	Marker softtool	default
Stimulus – [Start]	Alt + Shift + J	Stimulus softtool	Selects an input field corresponding to the start, stop, center or span of the active channel's current sweep type. Activates the corresponding tab on the "Stimulus" softtool.
Stimulus – [Stop]	Alt + Shift + K		
Stimulus – [Center]	Alt + Shift + F		
Stimulus – [Span]	Alt + Shift + H		
Channel – [Pwr Bw Avg]	Alt + Shift + L	Power Bw Avg softtool	default
Channel – [Sweep]	Alt + Shift + M	Sweep Softtool	default
Channel – [Cal]	Alt + Shift + P	Cal softtool	default
Channel – [Chan-nel Config]	Alt + Shift + O	Channel Config softtool	default
Channel – [Trigger]	Alt + Shift + R	"Trigger" softtool	opens the Trigger In tab
Channel – [Off-set Embed]	Alt + Shift + Q	Offset Embed softtool	default
System – [File]	Ctrl + O	File softtool	default

(Virtual) Hardkey	Keyboard Shortcut	Related Softtool	Action
System – [DUT] 2nd generation R&S ZNB models only (order nos. 1334.3330.xx)	Ctrl + P	Meas softtool	Opens the S-Parameter Wizard
System – [Print] (1st generation R&S ZNB and R&S ZNBT only)	Ctrl + P	File softtool	Opens the Print tab
System – [Display]	Alt + Shift + S	Display softtool	default
System – [Setup]	Alt + Shift + T	Setup softtool	default
System – [Applic]	Alt + Shift + N	Tools softtool	default
System – [Preset]	Alt + Shift + U	none	n.a.

5.2 Meas softtool

The "Meas" softtool allows you to select the quantities to be measured and displayed.

Access: Trace – [Meas]

Background information

For a detailed description of all measurement results of the R&S ZNB, refer to Section 4.3, "Measurement results", on page 116.

For information on efficient trace handling, see Section 3.3.5, "Handling diagrams, traces, and markers", on page 57.

Measurement Menu

The "Measurement Menu" combo-box at the top of the "Meas" softtool allows you to select between different measurement types.

- General-purpose measurements on non-frequency-converting DUTs
  See Section 5.2.1, "S-Parameters", on page 292
- Scalar mixer measurements
  See Section 5.2.2, "Mixer Measurements", on page 322
- Noise figure measurements on non-frequency-converting DUTs
  See Section 5.2.3, "Intermodulation", on page 333
- Intermodulation measurement on non-frequency-converting DUTs
  See Section 5.2.4, "Noise Figure", on page 348
- Time domain analysis of frequency domain data
  - Section 5.2.5, "Time Domain", on page 356
  - Section 5.2.6, "Extended Time Domain", on page 374
- Signal integrity measurements (user-defined or standardized)

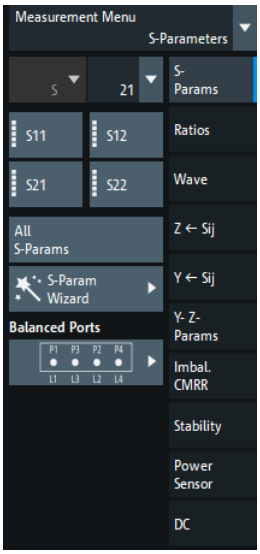
See Section 5.2.7, "Signal Integrity Apps", on page 410

- Measurement Add-Ins
  See Section 5.2.8, "External DLLs", on page 433

5.2.1 S-Parameters

The "S-Parameters" measurement mode offers general-purpose measurements on non-frequency-converting DUTs.

Access: Trace – [Meas] > "Measurement Menu: S-Parameters"



Access: Trace – [Meas] > "S-Parameters"

5.2.1.1 S-Params tab

Selects S-parameters as measured quantities. S-parameters are the basic measured quantities of a network analyzer. They describe how the DUT modifies a signal that is transmitted or reflected in forward or reverse direction. S-parameters (and derived quantities such as Y- and Z-parameters) fully characterize a linear DUT.

Background information

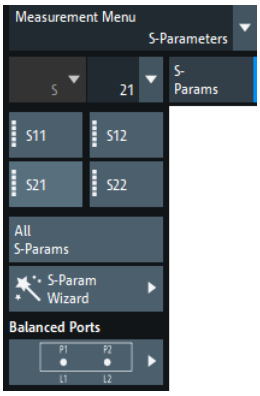
Refer to Section 4.3.1, "S-parameters", on page 117.

Port activation on demand

If a requested result involves disabled ports, but could be calculated if those ports were configured as single-ended ports with "logical port number = physical port number", then the required logical port configuration is performed automatically.

For example, with P1 assigned to L1 and P2 disabled, S21 could be measured if P2 would be assigned to L2.

Controls on the S-Params tab



Measurements: S-Parameters
Configures S-Parameter measurements.

S-Parameter (selector) ← Measurements: S-Parameters
Selects an S-parameters as a measured quantity for the active trace.

Single-ended (unbalanced) S-parameters are referred to as S<out>< in>, where <out> and <in> denote the output and input **logical** port numbers, respectively.

If entered manually, <out> and <in> must be expressed using the same number of digits. I.e. for <out>=10 and <in>=9 you can specify S<out>< in> as S1009 or S010009, but not as S109.

In presence of balanced ports, standard S-parameters are defined in the form S<m_out><m_in><out><in>, where output mode <m_out> and input mode <m_in> can be one of:

- d (differential, balanced)
- c (common, balanced)
- s (single-ended, unbalanced)

It is also possible to display "raw" single-ended S-parameters within the same channel: after setting the S-Parameter type to "S" any pair of (used) **physical** ports can be selected.

Remote command:

```
CALCulate<Ch>:PARameter:MEASure
CALCulate<Ch>:PARameter:MEASure:SENDed
CALCulate<Ch>:PARameter:SDEFine
CALCulate<Ch>:PARameter:SDEFine:SENDed
```

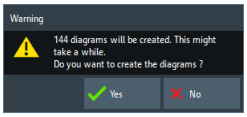
S<out><in> ← Measurements: S-Parameters
Selects one of the four elements of the standard 2-port S-parameters as a measured quantity for the active trace.

Remote command:

```
CALCulate<Ch>:PARameter:MEASure "<Trace_Name>", "S11" | "S12" |
"S21" | "S22"
CALCulate<Ch>:PARameter:SDEFine "<Trace_Name>", "S11" | "S12" |
"S21" | "S22"
```

All S-Params ← Measurements: S-Parameters
With the active logical ports distributed to m DUTs with n_1, ..., n_m ports, respectively, the analyzer creates n_1^2 + ... + n_m^2 diagrams and displays the full set of S-parameters, one in each diagram. The diagrams are arranged as an n x n grid.

If "All S-Params" would result in more than 100 diagrams being created, a warning dialog is opened that lets you cancel the diagram creation.



Remote command:
`CALCulate<Ch>:PARameter:SDEFine`
`CALCulate<Ch>:PARameter:DEFine:SGRoup`

S-Param Wizard ← Measurements: S-Parameters
Opens a dialog that guides you through the setup of a standard multi-port S-parameter measurement in a frequency sweep. See "S-Parameter Wizard" on page 294.

Dialog and opener button are not available if a switch matrix is configured.

Balanced Ports
Opens a dialog that allows you to enable/disable physical ports and to define logical ports (balanced or unbalanced) in the active channel. See "Balanced Ports dialog" on page 296.

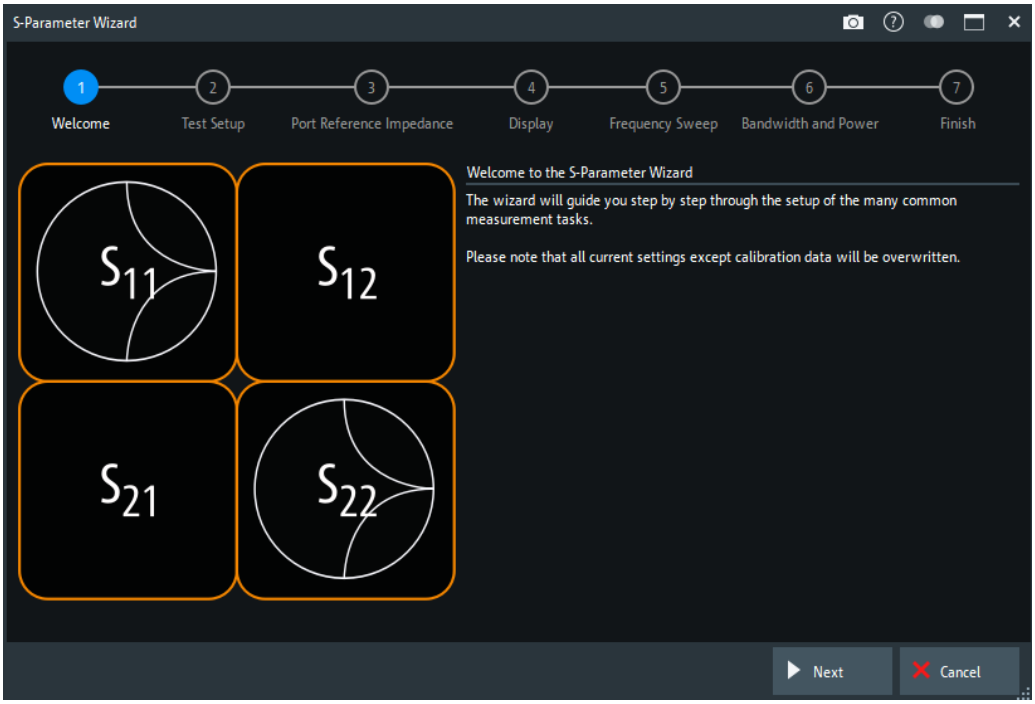
Note: If either multiple DUTs are configured (see Section 5.12.2.3, "Define Parallel Measurement dialog", on page 694) or the "Fixture Simulator" is disabled for the related channel (see "Fixture Simulator" on page 672), the "Balanced Ports" button is inactive (grayed out).

S-Parameter Wizard

The "S-Parameter Wizard" guides you through the setup of a standard S-parameter measurement in a frequency sweep.

This dialog is not available for R&S ZNBT, or if a switch matrix is configured.

Access: Trace – [Meas] > "S-Parameters" > "S-Params" > "S-Param Wizard"



The wizard proceeds in the following steps:

1. **Select the test setup.**
   Choose the port setup of the analyzer according to the port configuration of your DUT. Then connect the DUT to the selected analyzer ports.
   This step corresponds to the "Predefined Config" tab of the "Balanced Ports" dialog; see "Predefined Config tab" on page 297.

2. **Define the port impedances.**
   Assign reference impedances to all physical and balanced test ports selected in the previous step. The reference impedances can be complex.
   This step corresponds to the "Reference Impedance" tab of the "Balanced Ports" dialog; see "Reference Impedance tab" on page 300.

   Tip: The default reference impedance of the physical analyzer ports is Z_0 = 50 Ω. The default reference impedances for balanced ports are derived hereof. You do not need to change this value unless you want to renormalize the port impedances; see Section 4.3.2, "Reference impedances", on page 119.

3. **Select the measurement parameters and the diagram areas.**
   Depending on the test setup selected in step 1, the wizard offers different sets of S-parameters. Each S-parameter trace is displayed in a separate diagram. For transmission parameters, the wizard always selects dB magnitude format. For reflection parameters, you can select between Smith or dB magnitude format.

   Tip: You can always use the "Back" button to return to previous wizard steps and modify your settings.

4. **Select the sweep settings.**

Lets you choose the frequency range, the number of points per sweep, and the frequency sweep type.

5. **Select the measurement bandwidth and source power.**
   Choose a typical measurement bandwidth and one of three typical source power values. A smaller measurement bandwidth increases the dynamic range but slows down the measurement. If necessary, select a smaller source power to protect the input port of the analyzer.

   Note:
   The predefined bandwidths and source powers have been selected according to the following criteria:
   - The large measurement bandwidth ("Fast Sweep") ensures that the noise of an S_{21} trace at minimum source power and 0 dB attenuation is smaller than 0.1 dB.
   - The default source power for a passive DUT ensures that the analyzer receiver is in its linear range (no compression) if a passive DUT with 0 dB attenuation is measured. The default source powers for active DUTs ensure no compression if an active DUT with 20 dB or 40 dB gain is measured.
     If the actual gain of the DUT is higher than 50 dB, then the default source power of -40 dB is still too high. It must be changed after finishing the wizard.

6. **Perform a calibration (optional).**
   Allows you to perform a calibration. The "S-Parameter Wizard" proposes a full n-port calibration of the test ports selected in step 1. Automatic calibration is also possible (if a calibration unit is available).

   Tip:
   You can skip the calibration step (select "Finish now without Calibration") if one of the following applies:
   - A valid calibration is already assigned to the active channel
   - You want to apply a valid calibration stored in the cal pool.
   - You do not want to use a calibration, e.g. because the factory calibration is accurate enough for your measurement.

Instrument reset

To obtain a predictable result, the measurement wizard has to reset all settings except the current calibration data. Store your recall set if you do not want to lose the current configuration.

Balanced Ports dialog

The "Balanced Ports" dialog allows you to enable/disable physical ports and to define logical ports (balanced or unbalanced) in the active channel.

Access: Trace – [Meas] > "S-Parameters" > "S-Params" > "Balanced Ports"

Background information

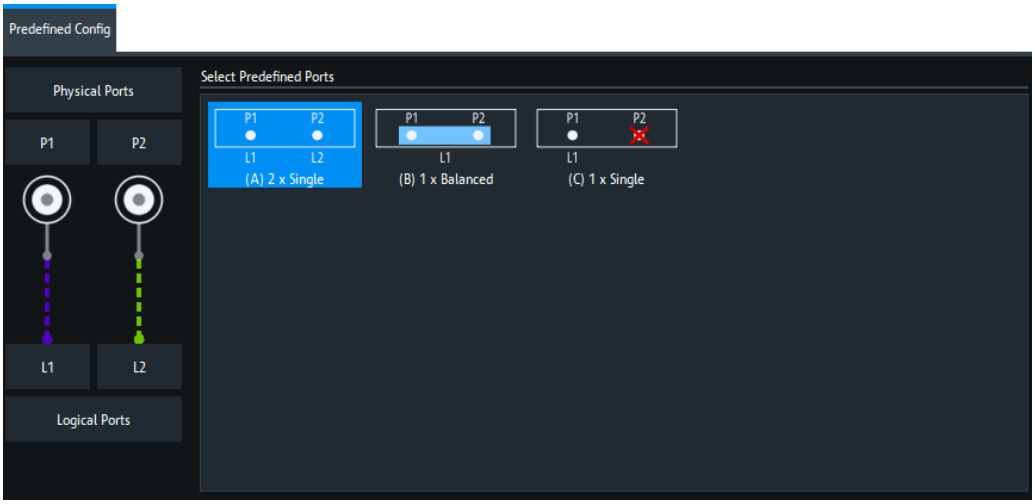
Refer to the following sections:

- Section 4.3.6, "Unbalance-balance conversion", on page 129
- Section 4.3.6.1, "Balanced port configurations", on page 130
- Section 4.3.2, "Reference impedances", on page 119

Predefined Config tab

The "Predefined Config" tab of the "Balanced Ports" dialog provides the most commonly used logical port configurations of the analyzer.

This tab is hidden if a switch matrix is configured.



The port configurations are arranged in the list to the right. The resulting port assignment is shown on the left-hand side of the "Predefined Config" tab.

- For a single-ended port, the diagram shows a single line between the physical test port and the logical port.
- For a balanced port, two physical ports are combined to form a single logical port.
- For unused ports, the physical port is crossed out; no logical port number is assigned.

Select Predefined Port

Allows you to select from a set of predefined logical port configurations.

The available configurations depend on the number of analyzer ports.

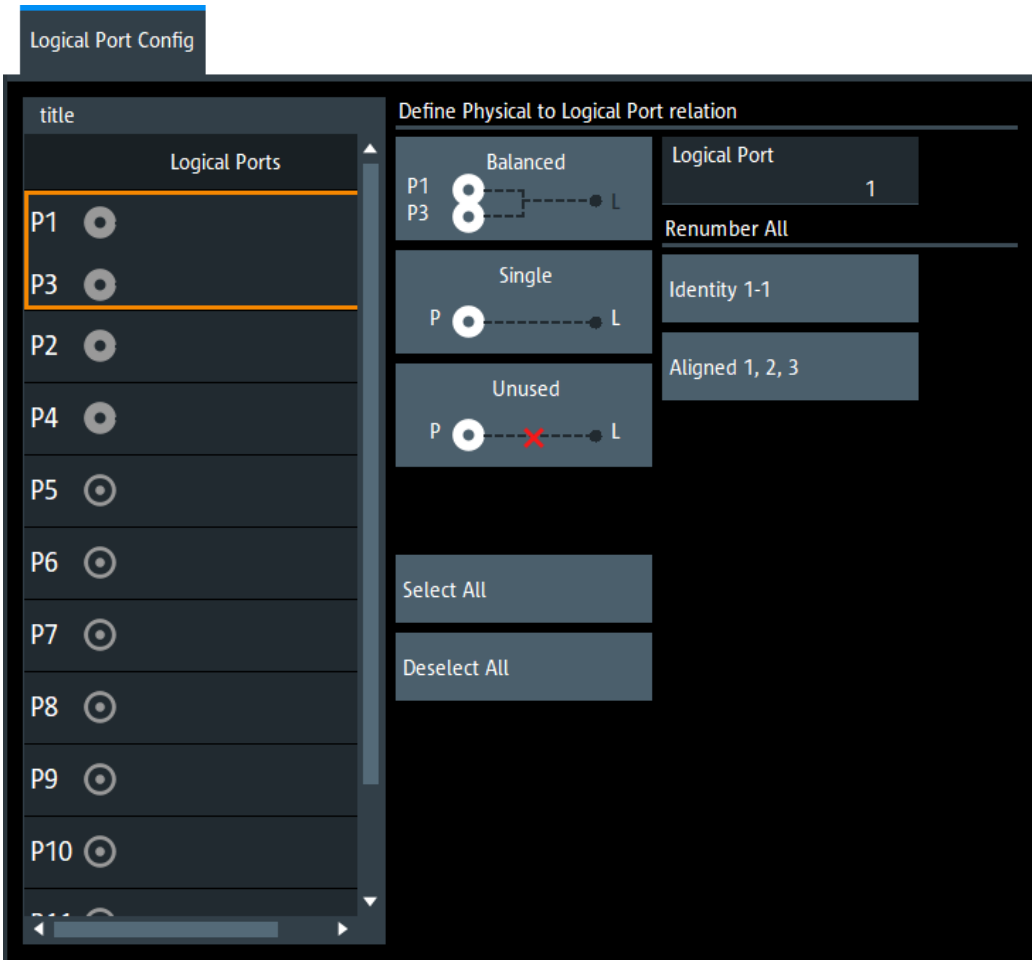
Remote command:

```
SOURce<Ch>:LPORt<LogPt>
SOURce<Ch>:LPORt<LogPt>:CLEar
```

Logical Port Config tab

The "Logical Port Config" tab of the "Balanced Ports" dialog allows you to configure logical ports (balanced or unbalanced) and to disable unused physical ports in the active channel.

This tab is only visible if a switch matrix is configured



Define Physical to Logical Port Relation
Allows you to define balanced, single-ended, and unused ports and provides functions for (re)numbering the resulting logical ports.

In principle, it is possible to combine any pair of two physical analyzer ports into a balanced port. With n test ports a maximum of n/2 (n even) or (n – 1)/2 (n odd) logical ports is supported.

- To define a balanced port, select two physical ports and tap "Balanced".
- To dissolve balanced ports, select them and tap "Single".
- To exclude logical ports from the measurement, select them and tap "Unused".
- To assign a number to a logical port, select it and enter a new "Logical Port" number in the corresponding field

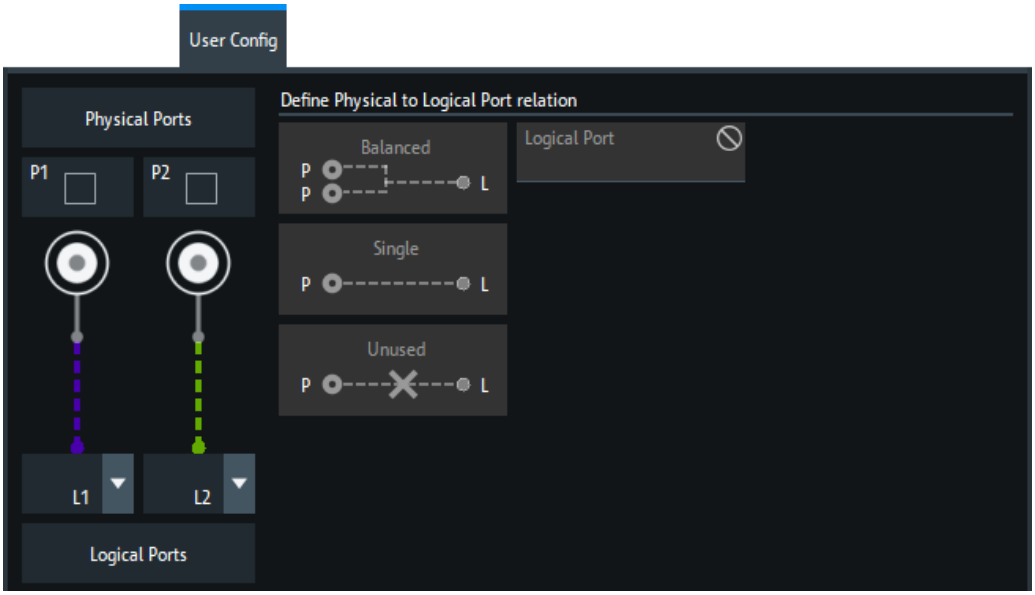
- To number the logical ports in line with the physical ports, select "Identity 1-1" For balanced ports, the lower of the two physical port numbers is selected.
- To number the logical ports consecutively from top to bottom, select "Aligned 1,2,3..."

Remote command:
SOURce<Ch>:LPORt<LogPt>
SOURce<Ch>:LPORt<LogPt>:CLEar

User Config tab

The "User Config" tab of the "Balanced Ports" dialog defines a new balanced port configuration.

If a switch matrix is configured, this tab is replaced by the Logical Port Config tab tab.



Physical Ports / Logical Ports
Allows you to renumber logical ports.

Remote command:
SOURce<Ch>:LPORt<LogPt>
SOURce<Ch>:LPORt<LogPt>:CLEar

Define Physical to Logical Port Relation
Allows you to define balanced, single-ended, and unused ports. In principle, it is possible to combine any pair of two physical ports into a balanced port.

- To define a balanced port, select two physical ports and tap "Balanced".
- To dissolve a balanced port, select it and tap "Single".
- To exclude a physical port from the measurement, select the port and tap "Unused".

Furthermore, provides functions for renumbering the logical ports.

Remote command:
SOURce<Ch>:LPORt<LogPt>
SOURce<Ch>:LPORt<LogPt>:CLEar

Reference Impedance tab

The "Reference Impedance" tab of the "Balanced Ports" dialog allows you to define (or redefine) the impedances of the logical ports.

Background information

Refer to Section 4.3.2, "Reference impedances", on page 119.

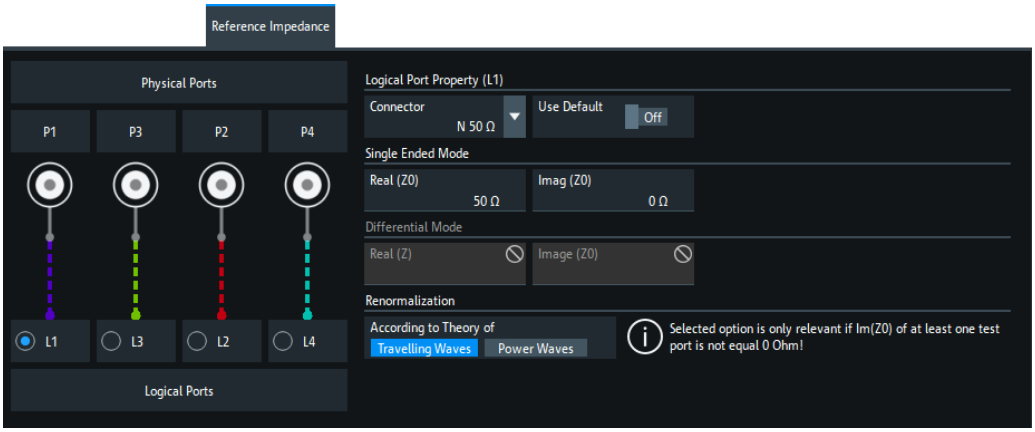


Figure 5-1: Reference Impedance Tab (4 physical ports)

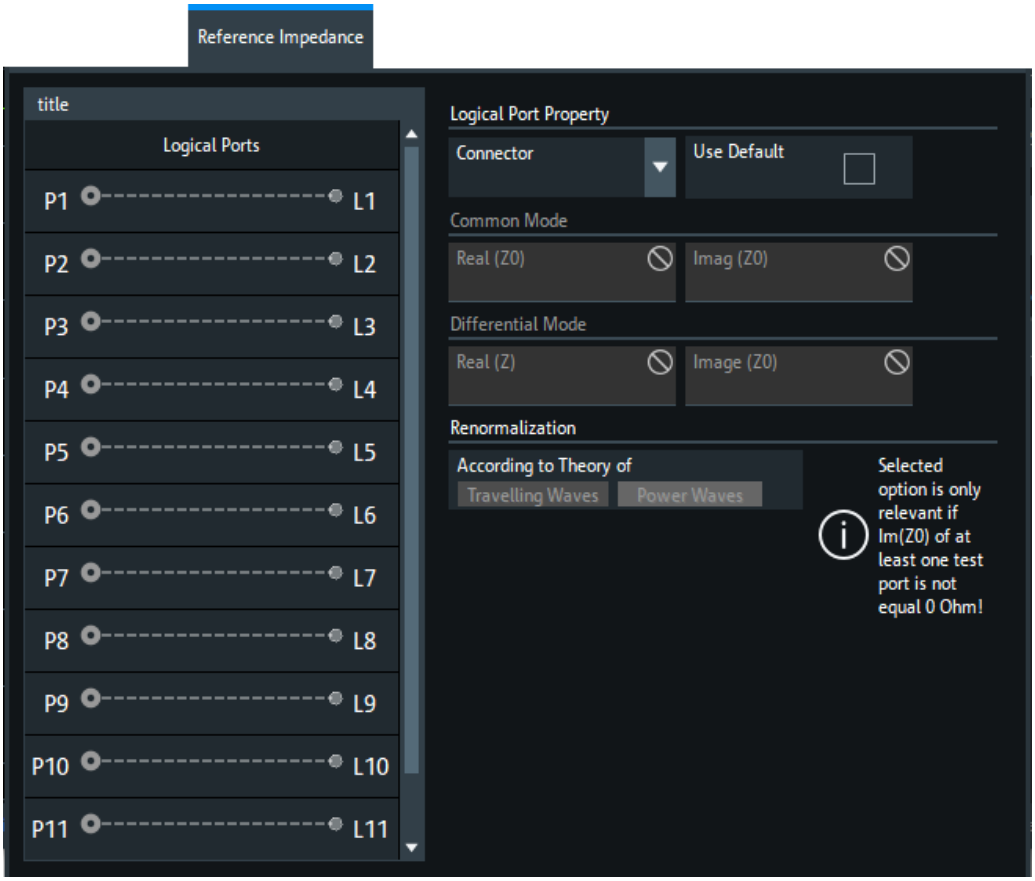
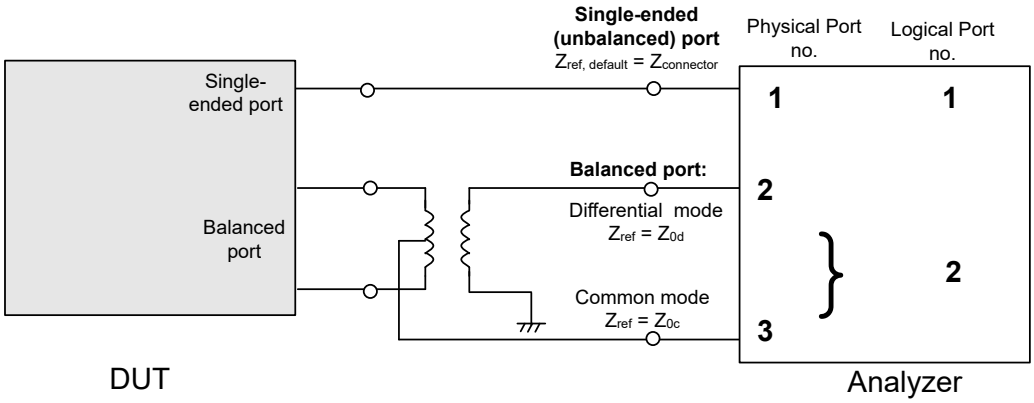


Figure 5-2: Reference Impedance Tab (>4 physical ports)

By default, the reference impedance of a physical port is set to the reference impedance of the connector type assigned to the port. However, it can be defined as an arbitrary complex value (renormalization of port impedances). By changing the reference impedance, it is possible to convert the measured values at 50 Ω (75 Ω) into values at arbitrary port impedances.

For balanced ports, it is possible to define separate complex reference impedances for differential and for common mode.



Single Ended Mode / Common Mode / Differential Mode
Defines arbitrary reference impedances.

"Single Ended Mode" is available for single-ended logical ports only, "Common Mode" and "Differential Mode" impedances for balanced ports only.

The default values for the balanced port reference impedances are derived from the (real) default reference impedance Z_0 = 50 Ω of the (single-ended) physical analyzer ports:

- The default value for the differential mode is Z_{0d} = 100 Ω = 2·Z_0.
- The default value for the common mode is Z_{0c} = 25 Ω = Z_0/2.

Remote command:
```
[SENSe<Ch>:]PORT<PhyPt>:ZREFerence
[SENSe<Ch>:]LPORt<LogPt>:ZCOMmon
[SENSe<Ch>:]LPORt<LogPt>:ZDIFferent
```

Connector
Allows you to specify the connector type of the related physical port.

Remote command:
```
[SENSe<Ch>:]CORRection:COLLect:CONNection<PhyPt>
```

Use Default
Allows you to toggle between default and renormalized port impedance (or impedances) for the selected logical port and connector type.

Remote command:
```
[SENSe<Ch>:]LPORt<LogPt>:ZDEFault[:STATe]
```

Renormalization According to Theory of
Selects the waveguide circuit theory for renormalization. The conversion formulas of these theories only differ if the reference impedance of at least one test port has a non-zero imaginary part.

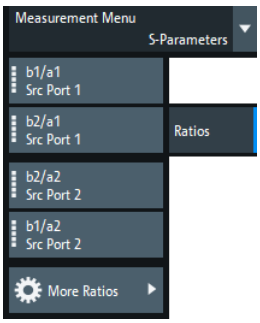
Refer to Section 4.3.2, "Reference impedances", on page 119.

Remote command:
```
CALCulate<Chn>:TRANsform:IMPedance:RNORmal
```

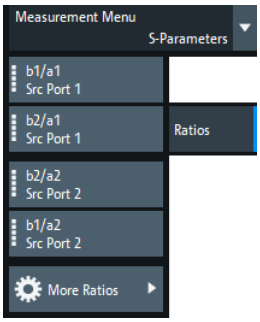
5.2.1.2 Ratios tab

Selects ratios of wave quantities as measured quantities.

Background information
Refer to Section 4.3.5, "Wave quantities and ratios", on page 126.

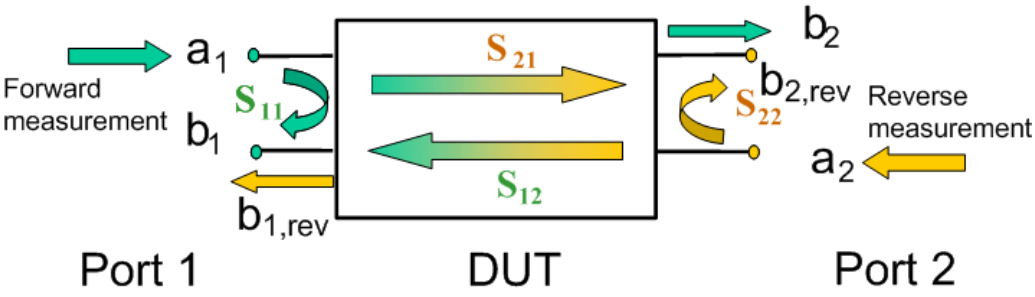


Controls on the Ratios tab



The "More Ratios" button opens the More Ratios dialog which allows to select an arbitrary ratio of wave quantities.

b<i> / a<j> Source Port <j>
Selects predefined complex ratios of the standard 2-port wave quantities a_1, a_2, b_1, and b_2.



The predefined wave quantities can all be obtained with the same test setup, where a 2-port DUT is connected between the analyzer ports 1 and 2. The stimulus signal is provided by the analyzer port 1 or 2 ("Source Port").

The predefined wave quantities correspond to the 2-port S-parameters:

- "b1/a1 Source Port 1" is the ratio of the wave quantities b_1 and a_1, measured at port 1. This ratio corresponds to the S-parameter S_{11} (input reflection coefficient).
- "b2/a1 Source Port 1" is the ratio of the wave quantities b_2 and a_1 and corresponds to the S-parameter S_{21} (forward transmission coefficient).
- "b2/a2 Source Port 2" is the ratio of the wave quantities b_2 and a_2, measured at port 2. This ratio corresponds to the S-parameter S_{22} (output reflection coefficient).

- "b1/a2 Source Port 2" is the ratio of the wave quantities b_1 and a_2 and corresponds to the S-parameter S_{12} (reverse transmission coefficient).

The analyzer can also measure arbitrary ratios for other source ports; see "More Ratios dialog" on page 304.

Tip: In the trace list, the source port is indicated in brackets. "b2/a1(P1)" denotes the ratio b_2/a_1 with source port 1.

Remote command:

```
CALCulate<Ch>:PARameter:MEASure "<Trace_Name>", "B2/A1" | ...
CALCulate<Ch>:PARameter:SDEFine "<Trace_Name>", "B2/A1" | ...
```

More Ratios dialog

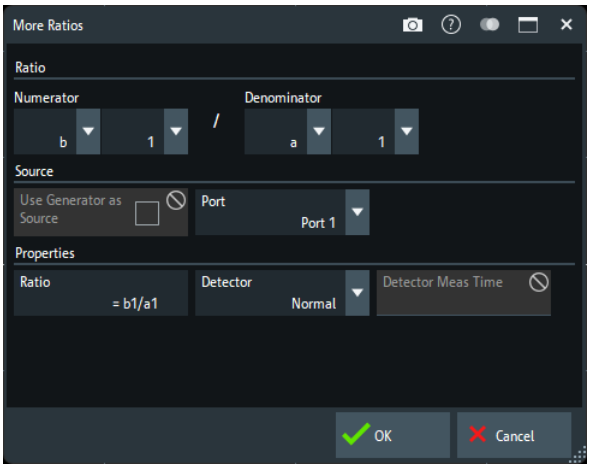
The More Ratios dialog allows you to select arbitrary ratios between wave quantities b_i, a_j as measured quantity. The ratios can be calculated with arbitrary source port and different detector settings.

Access: Trace – [Meas] > "S-Parameters" > "Ratios" > "More Ratios"

Background information

Refer to the following sections:

- Section 4.3.5.1, "Wave quantities", on page 126
- Section 4.3.5.2, "Ratios", on page 127



The notation for ratios follows the usual scheme of the vector network analyzer:

- The a-waves are the outgoing/transmitted waves at the analyzer's test ports.
- The b-waves are the incoming/measured waves.
- The source port for the stimulus signal must be specified in addition.
- The port number range covers all test ports of the analyzer.

Numerator

Selects the type (left pull-down list) and the port number assignment (right pull-down list) of the wave that forms the numerator of the ratio.

Remote command:
```
CALCulate<Ch>:PARameter:MEASure "<Trace_Name>", "B2/A1" | ...
CALCulate<Ch>:PARameter:SDEFine "<Trace_Name>", "B2/A1" | ...
```

Denominator
Selects the type (left pull-down list) and the port number assignment (right pull-down list) of the wave that forms the denominator of the ratio.

Remote command:
```
CALCulate<Ch>:PARameter:MEASure "<Trace_Name>", "B2/A1" | ...
CALCulate<Ch>:PARameter:SDEFine "<Trace_Name>", "B2/A1" | ...
```

Use Generator as Source
If selected, the "Port" control lists the external generators. Otherwise it lists the analyzer ports.

External generators must be configured explicitly in the "External Generators" dialog before they appear in the list. See also Section 4.7.29, "External generators", on page 269.

Port
Allows you to select the source for the stimulus signal.
- "None" or "Port <Pt>", if Use Generator as Source is not enabled
- "Gen <i>" if "Use Generator as Source" is enabled

Remote command:
```
CALCulate<Ch>:PARameter:MEASure "<Trace_Name>", "<Ratio>"
CALCulate<Ch>:PARameter:SDEFine "<Trace_Name>", "<Ratio>"
```

Detector
Selects the algorithm that is used to calculate the results points from the raw measurement data.

For details refer to Section 4.3.5.3, "Detector settings", on page 127.

Remote command:
```
CALCulate<Ch>:PARameter:MEASure
CALCulate<Ch>:PARameter:SDEFine
[SENSe<Ch>:]SWEep:DETector:TIME
```

5.2.1.3 Wave tab

Selects wave quantities as measured quantities.



Background information
Refer to Section 4.3.5, "Wave quantities and ratios", on page 126.

Controls on the Wave tab



The "More Wave Quantities" button opens the More Wave Quantities dialog which allows to select an arbitrary wave quantity, e.g. for different source ports or higher port numbers.

a<i> Source Port <i>, b<j> Source Port <i>
Selects one of the standard 2-port wave quantities a_i, b_j for different source ports.

The predefined wave quantities are obtained with different source ports. "a1 Source Port 1", "b1 Source Port 1" and "b1 Source Port 2" are measured at Port 1 of the analyzer. "a2 Source Port 2, b2 Source Port 1" and "b2 Source Port 2" are measured at Port 2 of the analyzer.

- "a1 Source Port 1" is the wave transmitted at physical port 1. In a standard S-parameter measurement, this wave is fed to the input port (port 1) of the DUT (forward measurement).
- "b1 Source Port 1" is the wave received at physical port 1. In a standard S-parameter measurement, this wave is reflected at port 1 of the DUT (forward measurement).
- "b2 Source Port 1" is the wave received at physical port 2. In a standard S-parameter measurement, this wave is transmitted at port 2 of the DUT (forward measurement).
- "a2 Source Port 2" is the wave transmitted at physical port 2. In a standard S-parameter measurement, this wave is fed to the output port (port 2) of the DUT (reverse measurement).
- "b1 Source Port 2" is the wave received at physical port 1. In a standard S-parameter measurement, this wave is transmitted at port 2 of the DUT (reverse measurement).
- "b2 Source Port 2" is the wave received at physical port 2. In a standard S-parameter measurement, this wave is fed to the output port (port 2) of the DUT (reverse measurement).

Tip: In the trace list, the source port is indicated in brackets. For example, "a1(P1)" denotes the wave a_1 with source port 1.

The analyzer can also measure arbitrary wave quantities for other source ports; see "More Wave Quantities dialog" on page 307.

Remote command:
```
CALCulate<Ch>:PARameter:MEASure "<Trace_Name>", "A1" | ...
CALCulate<Ch>:PARameter:SDEFine "<Trace_Name>", "A1" | ...
```

More Wave Quantities dialog

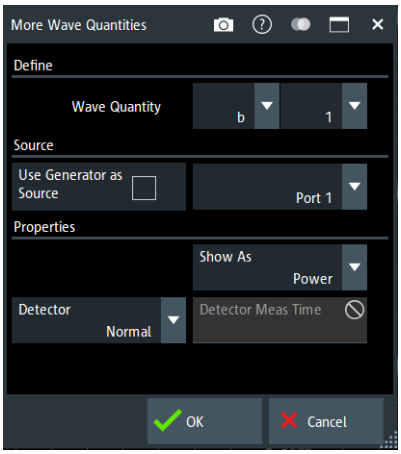
The "More Wave Quantities" dialog provides arbitrary wave quantities with arbitrary source ports as measured quantities. All wave quantities can be calculated with different detector settings.

Access: Trace – [Meas] > "S-Parameters" > "Wave" > "More Wave Quantities"

Background information

Refer to the following sections:

- Section 4.3.5.1, "Wave quantities", on page 126
- Section 4.3.5.2, "Ratios", on page 127



The notation for wave quantities follows the usual scheme of the vector network analyzer:

- The a-waves are the outgoing/transmitted waves at the analyzer's test ports.
- The b-waves are the incoming/measured waves.
- The source port for the stimulus signal must be specified in addition.
- The port number range covers all test ports of the analyzer.

Wave Quantity
Selects the type (left pull-down list) and the port number assignment (right pull-down list) of the wave quantitiy.

Remote command:
```
CALCulate<Ch>:PARameter:MEASure "<Trace_Name>", "A1" ...
CALCulate<Ch>:PARameter:SDEFine "<Trace_Name>", "A1" ...
```

Use Generator as Source
If enabled, the "Port" control lists the configured external generators. Otherwise it lists the analyzer ports.

To configure external generators, use the "External Generators" dialog. See Section 4.7.29, "External generators", on page 269.

Source > Port
Selects the source for the stimulus signal. Depending on the state of the "Use Generator as Source" flag either analyzer ports ("Port 1" to "Port n") or external generators ("Gen 1", "Gen 2", ...) can be selected.

The analyzer places no restriction on the combination of source ports and port numbers of the measured wave quantity. For example, it is possible to measure a_2 while port 1 is the source port (e.g. to estimate the directivity of the internal test set's directional element).

Remote command:
```
CALCulate<Ch>:PARameter:MEASure "<Trace_Name>", "A1" ...
CALCulate<Ch>:PARameter:SDEFine "<Trace_Name>", "A1" ...
```

Show as
Selects the physical unit of the displayed trace. It is possible to display the measured "Voltage" V or to convert it into a power value P according to the formula

$P = V^2 / Re(Z_0)$.

Z_0 denotes the reference impedance of the source port (for wave quantities a_n) or of the receive port (for wave quantities b_n). The reference impedances are defined in the "Balanced Ports" dialog; see "Balanced Ports dialog" on page 296.

Remote command:
```
CALCulate<Chn>:FORMat:WQUType
```

Detector
Selects the algorithm that is used to calculate the results points from the raw measurement data.

For details refer to Section 4.3.5.3, "Detector settings", on page 127.

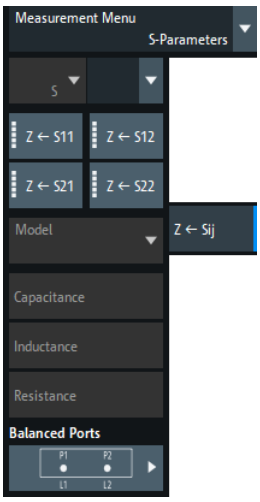
Remote command:
```
CALCulate<Ch>:PARameter:MEASure
CALCulate<Ch>:PARameter:SDEFine
[SENSe<Ch>:]SWEep:DETector:TIME
```

5.2.1.4 Z←Sij tab

Selects converted impedances, capacitances, inductances or resistances as measured quantities. The converted impedances are calculated from the measured S-parameters, the converted capacitances, inductances and resistances from the converted impedances.

Background information
Refer to the following sections:

- Section 4.3.3, "Impedance parameters", on page 121
- Section 4.3.3.1, "Converted impedances", on page 121



Z←S<out><in> selector
Selects a converted impedance parameter as a measured quantity for the active trace. For an n-port vector network analyzer, the pull-down list provides the full set of n^2 impedance parameters.

Converted impedance parameters are expressed as $Z \leftarrow S_{<out><in>}$, where <out> and <in> denote the output and input port numbers of the DUT.

Remote command:

```
CALCulate<Ch>:PARameter:MEASure "<Trace_Name>", "Z-S11" |
"Z-S12" ...
CALCulate<Ch>:PARameter:SDEFine "<Trace_Name>", "Z-S11" |
"Z-S22" | ...
```

Z←S<out><in> softkeys
Selects the 2-port converted impedance parameters. The parameters describe the impedances of a 2-port DUT, obtained in forward and reverse transmission and reflection measurements:

- Z_{11} is the input impedance at port 1 of a 2-port DUT that is terminated at port 2 with the reference impedance Z_0 (matched-circuit impedance measured in a forward reflection measurement).
- Z_{22} is the input impedance at port 2 of a 2-port DUT that is terminated at port 1 with the reference impedance Z_0 (matched-circuit impedance measured in a reverse reflection measurement).
- Z_{12} and Z_{21} denote the forward and reverse converted transfer impedances, respectively.

Use the Smith chart to obtain an alternative, graphical representation of the converted impedances in a reflection measurement.

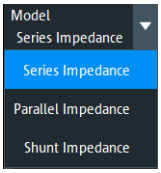
Tip: Use the "Y- Z-Params" tab to measure Z-parameters including the transfer parameters.

Remote command:

```
CALCulate<Ch>:PARameter:MEASure "<Trace_Name>", "Z-S11" |
"Z-S12" | "Z-S21" | "Z-S22"
CALCulate<Ch>:PARameter:SDEFine "<Trace_Name>", "Z-S11" |
"Z-S12" | "Z-S21" | "Z-S22"
```

Model

In case a transmission parameter is selected, you can choose between a series (default), a parallel, and a shunt impedance calculation.



Remote command:

```
CALCulate<Ch>:PARameter:MEASure "<Trace_Name>", "Z-S12" |
"Z-S12SER" | "Z-S12PAR" | ...
CALCulate<Ch>:PARameter:SDEFine "<Trace_Name>", "Z-S12" |
"Z-S12SER" | "Z-S12PAR" | ...
CALCulate<Ch>:PARameter:MEASure "<Trace_Name>", 'SHUNT-S12' |
'SHUNT-S13 | ... | 'SHUNT-S21' | 'SHUNT-S23' | ...
CALCulate<Ch>:PARameter:SDEFine "<Trace_Name>", 'SHUNT-S12' |
'SHUNT-S13 | ... | 'SHUNT-S21' | 'SHUNT-S23' | ...
```

Capacitance/Inductance/Resistance

Converted C-, L- and R-parameters are calculated from the converted impedance parameters Z_{ij} using the formulae:

$C_{ij} = -1/Im(Z_{ij}) \cdot 2\pi f$

$L_{ij} = Im(Z_{ij})/2 \cdot \pi f$

$R_{ij} = Real(Z_{ij})$

Remote command:

```
CALCulate<Ch>:PARameter:MEASure "<Trace_Name>", "C-S12" |
"C-S12SER" | "C-S12PAR" | "C-S12SHUNT" | ... | "L-S12" |
"L-S12SER" | "L-S12PAR" | "L-S12SHUNT" | ... | "R-S12" |
"R-S12SER" | "R-S12PAR" | "R-S12SHUNT" | ...
CALCulate<Ch>:PARameter:SDEFine "<Trace_Name>", "C-S12" |
"C-S12SER" | "C-S12PAR" | "C-S12SHUNT" | ... | "L-S12" |
"L-S12SER" | "L-S12PAR" | "L-S12SHUNT" | ... | "R-S12" |
"R-S12SER" | "R-S12PAR" | "R-S12SHUNT" | ...
```

Balanced Ports...

Opens a dialog to configure the logical ports of the analyzer.

See "Balanced Ports dialog" on page 296.

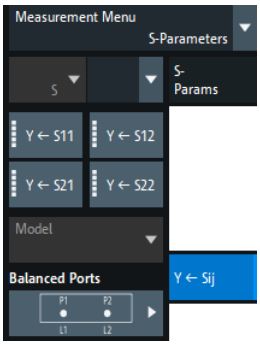
5.2.1.5 Y←Sij tab

Selects converted admittances as measured quantities. The admittances are calculated from the measured S-parameters.

Background information

Refer to the following sections:

- Section 4.3.4, "Admittance parameters", on page 125
- Section 4.3.4.1, "Converted admittances", on page 125



Y←S<out><in> selector

Selects a converted admittance parameter as a measured quantity for the active trace. For an n-port vector network analyzer, the pull-down list provides the full set of n^2 admittance parameters.

Converted admittance parameters are expressed as $Y{\leftarrow}S_{<out><in>}$, where <out> and <in> denote the output and input port numbers of the DUT.

Remote command:

```
CALCulate<Ch>:PARameter:MEASure "<Trace_Name>", "Y-S11" |
"Y-S12" ...
CALCulate<Ch>:PARameter:SDEFine "<Trace_Name>", "Y-S11" |
"Y-S22" | ...
```

Y←S<out><in> softkeys

Selects the 2-port converted admittance parameters. The parameters describe the admittances of a 2-port DUT, obtained in forward and reverse transmission and reflection measurements:

- Y_{11} is the input admittance at port 1 of a 2-port DUT that is terminated at port 2 with the reference impedance Z_0 (matched-circuit admittance measured in a forward reflection measurement).
- Y_{22} is the input admittance at port 2 of a 2-port DUT that is terminated at port 1 with the reference impedance Z_0 (matched-circuit admittance measured in a reverse reflection measurement).
- Y_{12} and Y_{21} denote the forward and reverse converted transfer admittances, respectively.

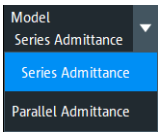
Use the Smith chart to obtain an alternative, graphical representation of the converted impedances in a reflection measurement.

Tip: Use the "Y- Z-Params" tab to measure Y-parameters including the transfer parameters.

Remote command:

```
CALCulate<Ch>:PARameter:MEASure "<Trace_Name>", "Y-S11" |
"Y-S12" | "Y-S21" | "Y-S22"
CALCulate<Ch>:PARameter:SDEFine "<Trace_Name>", "Y-S11" |
"Y-S12" | "Y-S21" | "Y-S22"
```

Model
In case a transmission parameter is selected, you can choose between a series (default), and a parallel admittance calculation.



Remote command:

```
CALCulate<Ch>:PARameter:MEASure "<Trace_Name>", "Y-S12" |
"Y-S12SER" | "Y-S12PAR" | ...
CALCulate<Ch>:PARameter:SDEFine "<Trace_Name>", "Y-S12" |
"Y-S12SER" | "Y-S12PAR" | ...
```

Balanced Ports...
Opens a dialog to configure the logical ports of the analyzer.

See "Balanced Ports dialog" on page 296.

5.2.1.6 Y-Z-Params tab

Allows you to select Y- and Z-parameters as measured quantities. Both Y- and Z-parameters can serve as an alternative to S-parameters for characterizing a linear n-port network.

Background information
Refer to the following sections:

- Section 4.3.3, "Impedance parameters", on page 121
- Section 4.3.4, "Admittance parameters", on page 125
- Section 4.3.3.2, "Z-parameters", on page 124
- Section 4.3.4.2, "Y-parameters", on page 125



Y/Z-Parameter
Selects an Y-parameter or Z-parameter as a measured quantity for the active trace. For an n-port vector network analyzer, the pull-down list provides the full set of n^2 Y- and Z-parameters.

Y- and Z-parameters are expressed as $Y/Z_{<out><in>}$, where <out> and <in> denote the output and input port numbers of the DUT.

Remote command:
```
CALCulate<Ch>:PARameter:MEASure "<Trace_Name>", "Y11" |
"Z11" ...
CALCulate<Ch>:PARameter:SDEFine "<Trace_Name>", "Y11" |
"Z11" ...
```

Y11/Y12/Y21/Y22
Selects one of the 2-port Section 4.3.4.2, "Y-parameters", on page 125 as a measured quantity for the active trace. The Y-parameters describe the admittances of a DUT with output ports terminated in a short circuit (V = 0).

The four 2-port Y-parameters can be interpreted as follows:

- Y_{11} is the input admittance, defined as the ratio of the current I_1 to the voltage V_1, measured at port 1 (forward measurement with output terminated in a short circuit, $V_2 = 0$).
- Y_{21} is the forward transfer admittance, defined as the ratio of the current I_2 to the voltage V_1 (forward measurement with output terminated in a short circuit, $V_2 = 0$).
- Y_{12} is the reverse transfer admittance, defined as the ratio of the current I_1 to the voltage V_2 (reverse measurement with input terminated in a short circuit, $V_1 = 0$).
- Y_{22} is the output admittance, defined as the ratio of the current I_2 to the voltage V_2, measured at port 2 (reverse measurement with input terminated in a short circuit, $V_1 = 0$).

Remote command:
```
CALCulate<Ch>:PARameter:MEASure "<Trace_Name>", "Y11" | "Y12" |
"Y21" | "Y22"
CALCulate<Ch>:PARameter:SDEFine "<Trace_Name>", "Y11" | "Y12" |
"Y21" | "Y22"
```

Z11 / Z12 / Z21 / Z22
Selects one of the 2-port Section 4.3.3.2, "Z-parameters", on page 124 as a measured quantity for the active trace. The Z-parameters describe the impedances of a DUT with open output ports (I = 0).

The four 2-port Z-parameters can be interpreted as follows:

- Z_{11} is the input impedance, defined as the ratio of the voltage V_1 to the current I_1, measured at port 1 (forward measurement with open output, $I_2 = 0$).
- Z_{21} is the forward transfer impedance, defined as the ratio of the voltage V_2 to the current I_1 (forward measurement with open output, $I_2 = 0$).
- Z_{12} is the reverse transfer impedance, defined as the ratio of the voltage V_1 to the current I_2 (reverse measurement with open input, $I_1 = 0$).
- Z_{22} is the output impedance, defined as the ratio of the voltage V_2 to the current I_2, measured at port 2 (reverse measurement with open input, $I_1 = 0$).

Remote command:

```
CALCulate<Ch>:PARameter:MEASure "<Trace_Name>", "Z11" | "Z12" | "Z21" | "Z22"
CALCulate<Ch>:PARameter:SDEFine "<Trace_Name>", "Z11" | "Z12" | "Z21" | "Z22"
```

Balanced Ports...
Opens a dialog to define a balanced port configuration.

See Balanced Ports dialog.

5.2.1.7 Imbal. CMRR tab

Selects an imbalance or common mode rejection ratio (CMRR) as measured quantity.

The "Imbal. CMRR" measurements require at least two logical ports, at least one of them balanced.

Background information
Refer to the following sections:

- Section 4.3.6, "Unbalance-balance conversion", on page 129
- Section 4.3.6.3, "Imbalance and common mode rejection", on page 132

Controls on the Imbal. CMRR tab



The following buttons in the "Imbal. CMRR" tab open additional dialogs:

- "Imbalance Differential...": see "Imbalance Differential dialog" on page 316. This button is only enabled if a balanced and two single-ended logical ports are available.
- "Balanced Ports": see "Balanced Ports dialog" on page 296

Imbalance/CMRR
Selects an imbalance or CMRR parameter as a measured quantity for the active trace.

These parameters are expressed as "$Imb_{<out><in>}$" or "$CMRR_{<out><in>}$", where <out> and <in> denote the logical output and input port numbers of the DUT.

Remote command:

```
CALCulate<Ch>:PARameter:MEASure "<Trace_Name>", "IMB21" |
"IMB12" | "CMRR11" | "CMRR21" | "CMRR12" ...
CALCulate<Ch>:PARameter:SDEFine "<Trace_Name>", "IMB21" |
"IMB12" | "CMRR11" | "CMRR21" | "CMRR12" ...
```

Imb21/Imb12/CMRR21/CMRR12
Selects one of the standard 2-port imbalance or CMRR parameters as a measured quantity for the active trace. The buttons are available if either logical port 1 or logical port 2 (or both) is defined as a balanced port.

A yellow font color indicates that currently the other logical port number is undefined and the physical port with the same number is marked as unused. If you select it as the measured quantity, the port is automatically enabled as a single-ended port with "logical port number" = "physical port number".

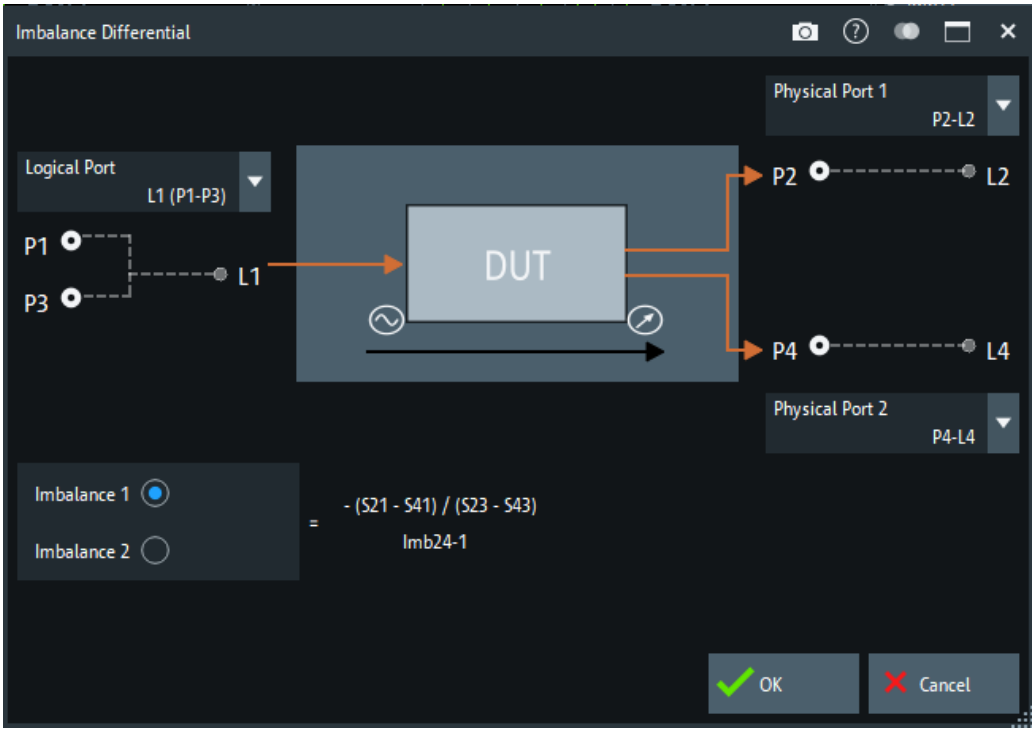
Remote command:

```
CALCulate<Ch>:PARameter:MEASure "<Trace_Name>", "IMB21" |
"IMB12" | "CMRR21" | "CMRR12"
CALCulate<Ch>:PARameter:SDEFine "<Trace_Name>", "IMB21" |
"IMB12" | "CMRR21" | "CMRR12"
```

Imbalance Differential dialog

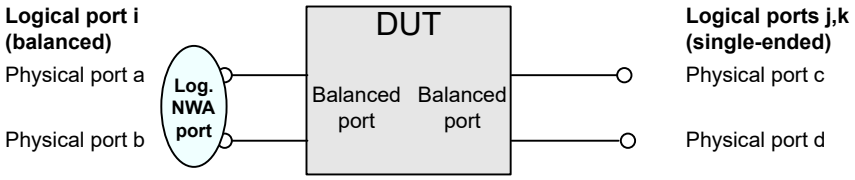
Allows you to measure the differential mode imbalance of a DUT with balanced input and output ports, connected to a balanced and two single-ended logical ports at the VNA.

Access: Trace – [Meas] > "S-Parameters" > "Imbal. CMRR" > "Imbalance Differential"



This dialog can only be opened, if a balanced and two single-ended logical ports are configured in the Balanced Ports dialog.

- The **Logical Port** on the left represents the balanced test port: any (active) balanced logical port can be selected
- Physical Port 1 and Physical Port 2 represent the two single-ended test ports: any two (active) single-ended logical ports can be selected
- The part below allows you to select the signal direction and hence the "Imbalance Result" to be calculated.



Imbalance 1 / Imbalance 2
Imbalance 1 selects the balanced logical port i as the input, the single-ended logical ports j and k as the output and calculates the following imbalance parameter:

$Imb_{jk\text{-}i} = -(S_{ca} - S_{da})/(S_{cb} - S_{db})$

Imbalance 2 selects the balanced logical port i as the output, the single-ended logical ports j and k as the input and calculates the following imbalance parameter:

$Imb_{i\text{-}jk} = -(S_{ac} - S_{ad})/(S_{bc} - S_{bd})$

Remote command:

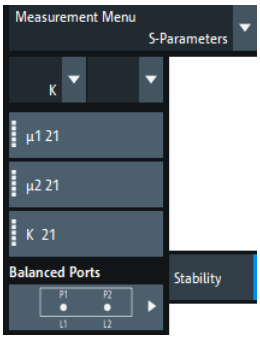
```
CALCulate<Ch>:PARameter:MEASure "<Trace_Name>", "IMB<i>-<j><k>"
| "IMB<j><k>-<i>"
CALCulate<Ch>:PARameter:SDEFine "<Trace_Name>", "IMB<i>-<j><k>"
| "IMB<j><k>-<i>"
```

5.2.1.8 Stability tab

Selects one of the three two port stability factors K, μ_1 or μ_2 as measured quantities. A typical application of stability factors is to assess the stability of an amplifier. Stability factors cannot be calculated in balanced port configurations.

Background information

Refer to Section 4.3.7, "Stability factors", on page 134.



Stability

Selects a stability factor as a measured quantity for the active trace. The stability factor calculation is based on 2-port reflection **and** transmission S-parameters so that the input and output port numbers must be different. The pull-down list contains all possible physical (single-ended) port combinations. For an analyzer with n ports, provides n * (n – 1) stability parameters.

Stability parameters are expressed as "$K_{<out><in>}$", "$\mu1_{<out><in>}$", and "$\mu2_{<out><in>}$", where <out> and <in> denote the logical output and input port numbers of the DUT.

Remote command:

```
CALCulate<Ch>:PARameter:MEASure "<Trace_Name>", "KFAC21" |
"MUF121" | "MUF221" | ...
CALCulate<Ch>:PARameter:SDEFine "<Trace_Name>", "KFAC21" |
"MUF121" | "MUF221" | ...
```

μ1 21/μ2 21/K 21

Selects one of the standard 2-port stability factors as a measured quantity for the active trace. These buttons are enabled if none of the logical ports 1 and 2 is defined as a balanced port.

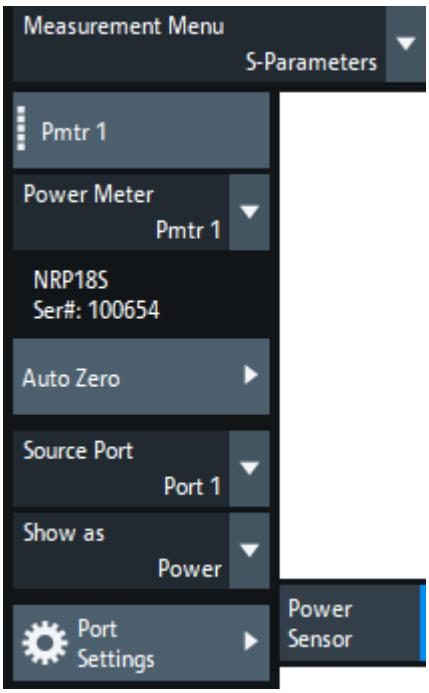
Remote command:
```
CALCulate<Ch>:PARameter:MEASure "<Trace_Name>", "MUF121" | "MUF221" | "KFAC21"
CALCulate<Ch>:PARameter:SDEFine "<Trace_Name>", "MUF121" | "MUF221" | "KFAC21"
```

Balanced Ports...
Opens a dialog to define a balanced port configuration.

See "Balanced Ports dialog" on page 296.

5.2.1.9 Power Sensor tab

Allows you to set up and perform measurements using external power sensors.



The standard test setup for a "Power Sensor" measurement involves one analyzer source port and a power sensor. The power sensor is connected to the VNA (e.g. to the analyzer's USB port) and provides scalar wave quantity results. See Section 4.7.28, "External power meters", on page 267.

Pmtr<i>
After a power meter was connected, configured, and selected, you can use this button to create a power meter trace in one of the available diagrams or in a new one.

Remote command:
```
CALCulate<Ch>:PARameter:MEASure 'TraceName', 'PmtrD1 | ...
CALCulate<Ch>:PARameter:SDEFine 'TraceName', 'PmtrD1 | ...
```

Power Meter
Shows a list of all power meters that have been properly configured. See "Configured Devices" on page 846.

The bordered label below displays the type and serial number of the selected power meter.

Remote command:
```
CALCulate<Ch>:PARameter:MEASure 'TraceName', 'PmtrD1 | ...
CALCulate<Ch>:PARameter:SDEFine 'TraceName', 'PmtrD1 | ...
```

Auto Zero
Initiates an automatic zeroing procedure of the selected power meter.

The power meter must be disconnected from the RF power; see Section 4.7.28.1, "Zeroing", on page 268. A message indicates that zeroing is finished.

Remote command:
```
SYSTem:COMMunicate:RDEVice:PMETer<Pmtr>:AZERo
```

Source Port
Selects one of the available test ports of the analyzer as a source of the stimulus signal.

Remote command:
```
CALCulate<Ch>:PARameter:MEASure 'TraceName', 'PmtrD1 | ...
CALCulate<Ch>:PARameter:SDEFine 'TraceName', 'PmtrD1 | ...
```

Show as
Selects the physical unit of the displayed trace. It is possible to display the measured "Voltage" V or convert it to a "Power" according to the formula

$P = V^2/Re(Z_0)$.

Z_0 denotes the reference impedance of the source port. The reference impedances are defined in the "Balanced Ports" dialog (see "Balanced Ports dialog" on page 296).

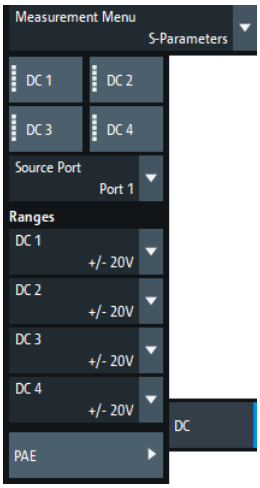
Remote command:
```
CALCulate<Chn>:FORMat:WQUType
```

Port Settings...
Opens the Port Settings dialog.

5.2.1.10 DC tab

Selects the ranges for the DC voltages fed to the DC INPUT connectors at the rear panel as measured quantities and opens the PAE measurement dialog. The DC input connectors are available as option R&S ZNB-B81 (see Section 4.7.26, "DC inputs", on page 264).

Controls on the DC tab



The "PAE" button opens the Power Added Efficiency dialog.

DC 1/DC 2/DC 3/DC 4/Source Port
Selects a DC INPUT and the drive port for the active trace.

The DC voltages "DC 1", ..., "DC 4" are measured at the rear panel connectors DC INPUT 1, ..., 4, respectively. For best accuracy, it is recommended to adjust the "Ranges" to the measured signal.

Remote command:

```
CALCulate<Ch>:PARameter:MEASure "<Trace_Name>", "DC1d1" | "DC2d1" | ...
CALCulate<Ch>:PARameter:SDEFine "<Trace_Name>", "DC1d1" | "DC2d1" | ...
```

Ranges
"DC 1", ..., "DC 4" in the "Ranges" section configure the measurement ranges of DC INPUT 1, ..., 4, respectively. For best accuracy, adjust the ranges to the measured signals. To protect the instrument, all ranges are pre-set to "+/- 20 V" (the widest range).

If the actual voltage of the input signal exceeds the selected range, the measurement is incorrect, and a DC overload message is displayed.

Remote command:

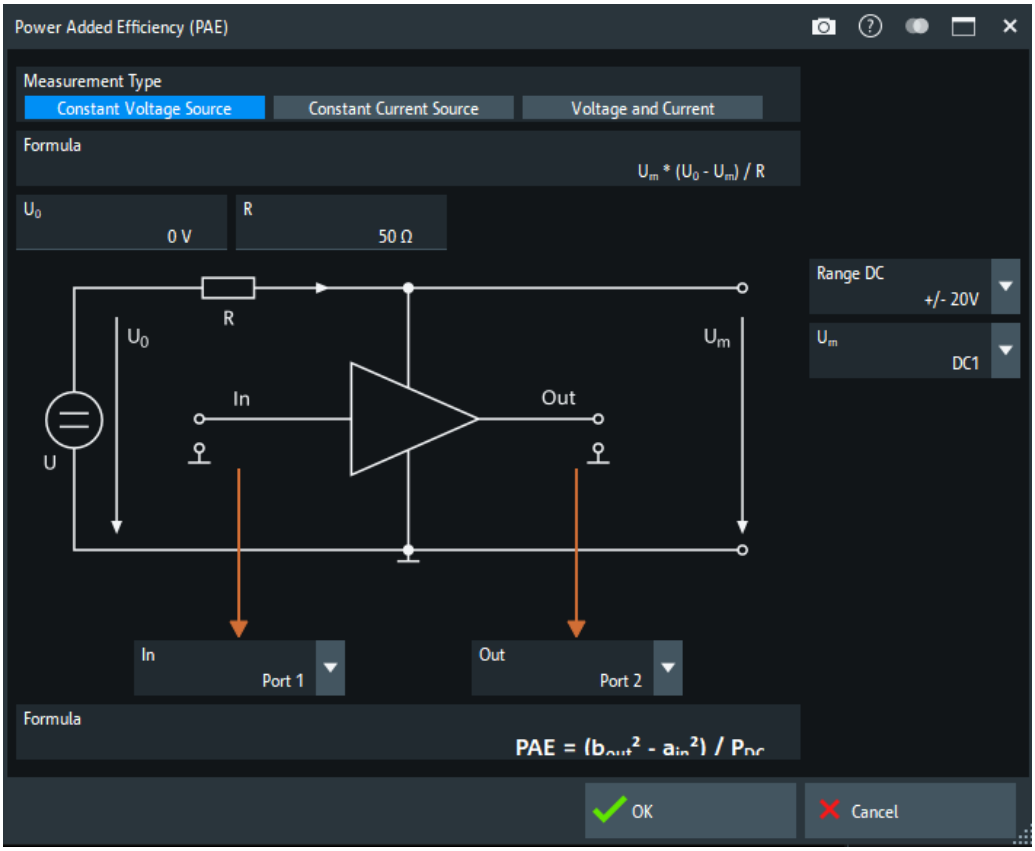
```
[SENSe<Ch>:]DC<DCInp>:RANGe
```

Power Added Efficiency (PAE) dialog

The "Power Added Efficiency (PAE)" dialog configures the measurement of the PAE of an active 2-port device.

Access: Section 5.2.1.10, "DC tab", on page 319 > "PAE"

Background information
Refer to section Section 4.7.26.1, "Power added efficiency", on page 265.



A PAE measurement involves the following steps:

1. Select the "Measurement Type", according to the properties of your DC power source.
2. Enter the parameters for the selected measurement type.
3. Establish the test setup (including the RF connections of the DUT and the DC INPUT connections) as shown in the circuit diagram.
4. Close the "Power Added Efficiency (PAE)" dialog and observe the result in the measurement diagram.

Measurement Type
Selects the PAE measurement type, depending on the DC power source in use:

- DC source providing a constant, known voltage at (possibly) variable current
- DC source providing a constant, known current at (possibly) variable voltage
- DC source with variable (unknown) current and power

The three measurement types involve different test setups, input parameters, and DC voltage measurements. For an overview, see Table 4-29.

Note: The measurement types for "Constant Voltage Source" and "Voltage and Current" require an additional non-zero, known precision resistor R.

Remote command:
`[SENSe<Ch>:]PAE:TYPE`

Formula
Shows the formula for the calculation of the DC supply power P_{DC}, depending on the selected Measurement Type. The parameters R, U_0, and/or I_0 must be entered in accordance with the properties with the DC power source and precision resistor in use.

Remote command:
```
[SENSe<Ch>:]PAE:PARameters:I
[SENSe<Ch>:]PAE:PARameters:R
[SENSe<Ch>:]PAE:PARameters:U
```

In/Out
Define the logical ports for the RF input and output signal, respectively.

Remote command:
```
CALCulate<Ch>:PARameter:MEASure 'TraceName', 'PAE21 | ...'
CALCulate<Ch>:PARameter:SDEFine 'TraceName', 'PAE21 | ...'
```

U_m/U_{m1}/U_{m2}/
Select the DC INPUT connectors used to determine the DC supply power.

How many of them are required depends on the selected Measurement Type.

Remote command:
```
[SENSe<Ch>:]PAE:DCINput:MAIN
[SENSe<Ch>:]PAE:DCINput:SECondary
```

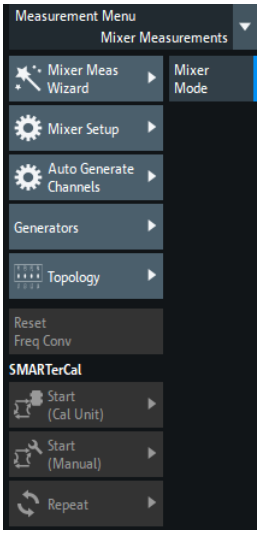
Range DC
Select the DC range of each DC INPUT connector in use according to the (estimated) input voltage range; see "DC 1/DC 2/DC 3/DC 4/Source Port" on page 320.

Remote command:
```
[SENSe<Ch>:]DC<DCInp>:RANGe
```

5.2.2 Mixer Measurements

Measurements on frequency-converting DUTs ("Mixer Measurements") require software option R&S ZNB-K4 (see Section 4.7.3, "Frequency conversion measurements", on page 233). If this option is not available, the "Measurement Menu" item "Mixer Measurements" and the corresponding tabs are not available.

Access: Trace – [Meas] > "Measurement Menu: Mixer Measurements"



5.2.2.1 Mixer Mode tab

Controls the measurement of the mixing products generated by a mixer under test (MUT) that is supplied with two independent RF signals. The scalar mixer mode is included in option R&S ZNB-K4. An extension to a two-stage mixer system is possible.

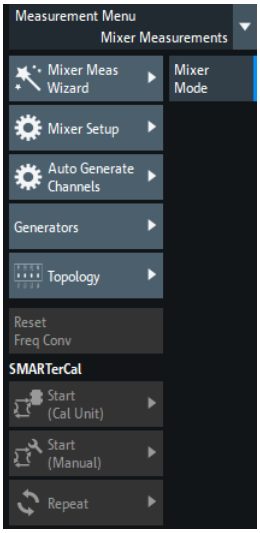
Two dedicated "SMARTerCal" wizards facilitate the channel calibration.

Background information

Refer to the following sections:

- Section 4.7.3.2, "Scalar mixer measurements", on page 234
- "Two-stage mixer measurements" on page 235

Controls on the Mixer Mode tab



Most of the "Mixer Mode" buttons are only enabled if the mixer measurement is set up properly. They open wizards or dialogs:

- **"Mixer Meas Wizard...":** See "Mixer Presetting wizard" on page 324
- **"Mixer Setup...":** See "Mixer Setup dialog" on page 326
- **"SMARTerCal" – "Start... (Cal Unit)":** see "SMARTerCal (Cal Unit) wizard for mixer mode" on page 330
- **"SMARTerCal" – "Start... (Manual)":** see "SMARTerCal (Manual) wizard for mixer mode" on page 332
- **"SMARTerCal" – "Repeat...":** launches the adequate "Mixer Meas"-specific wizard to repeat the currently active "SMARTerCal" (if available)

Reset Freq Conv
"Reset Freq Conv" disables all frequency conversion measurements in the active channel, including intermodulation measurements.

A "Mix" label in the channel list indicates that a mixer measurement is active.



Remote command:
`[SENSe<Ch>:]FREQuency:CONVersion`

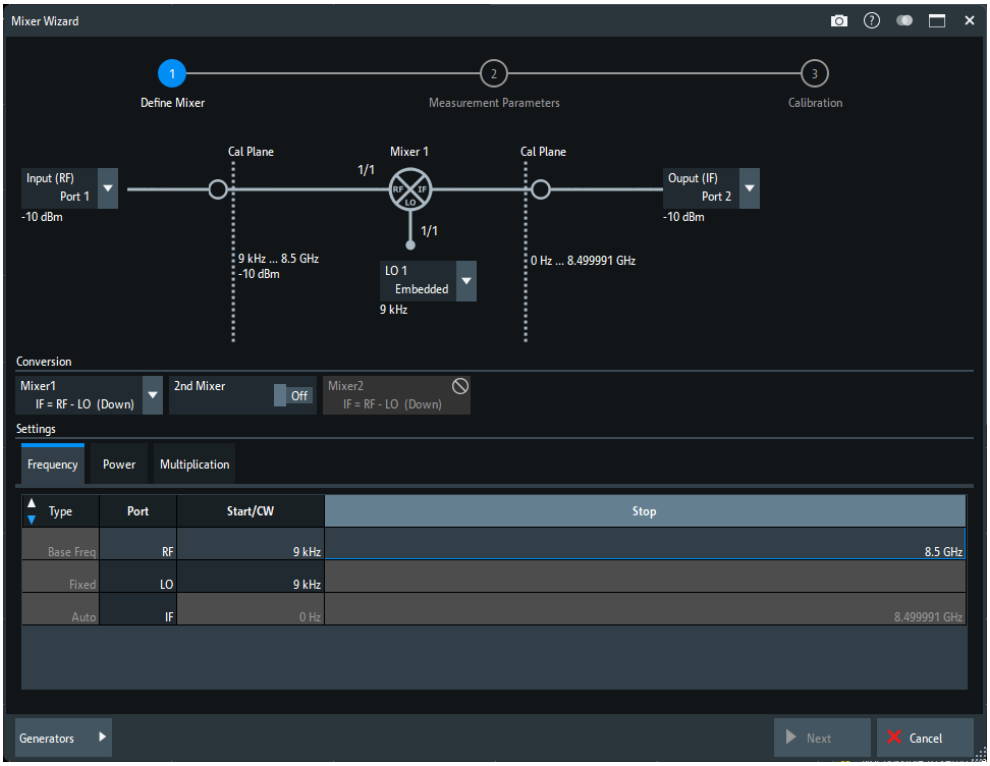
Mixer Presetting wizard

The "Mixer Presetting" wizard guides you through the setup of a mixer measurement.

Access: Channel – [Channel Config] > "Mixer Mode" > "Mixer Meas Wizard..."

The setup of the mixer measurement proceeds in 4 steps:

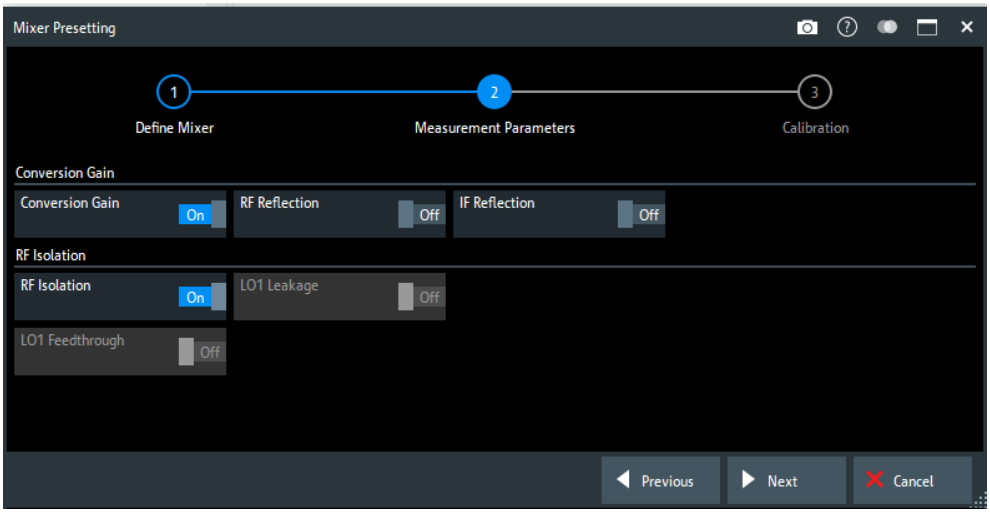
1. Define the mixer.
   See "Mixer Setup dialog" on page 326.



2. Select the mixer parameters to be measured. Depending on your selection, the required channels and traces are set up automatically; the active channel and its first trace will be renamed, all other channels and traces in the active recall set will be deleted.

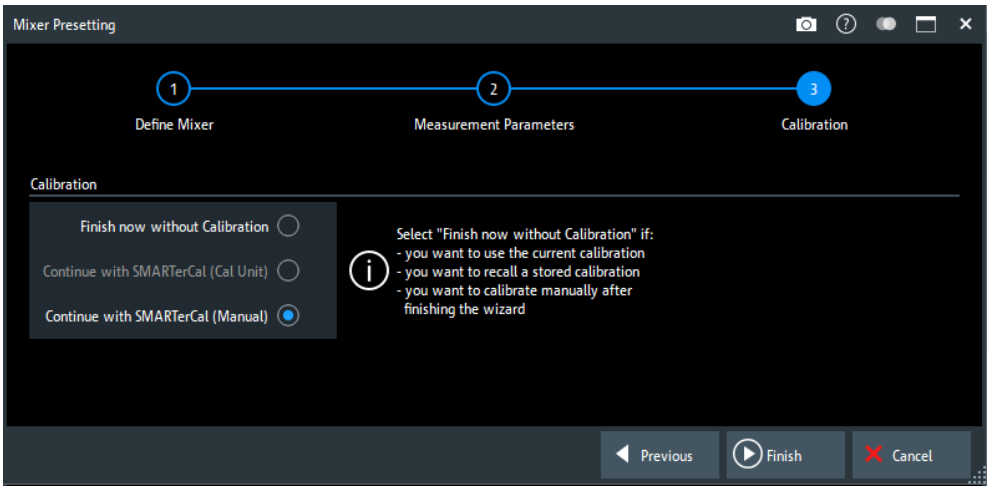
Table 5-2: Auto-created channels and traces

Measurement Parameter	Channel Name	Trace Name
Conversion Gain S<j><i>	Ch_M	Conv
RF Reflection S<i><i>	Ch_M	RF_Refl
IF Reflection S<j><j>	Ch_M	IF_Refl
RF Isolation S<j><i> [1)]	Ch_RF	RF_Isol
LO[m] Leakage S<i><k>	Ch_LO	LO_Leak
LO[m] Feedthrough S<j><k>	Ch_LO	LO_Thru
<i> = RF test port number, <j> = IF test port number,<k> = LO test port number [m] = Mixer number (only if 2 mixers are measured) [1)] measured w/o frequency conversion		



Note that LO parameters can only be measured if a VNA port was selected as the respective LO in the previous step. This in turn requires a second internal source (see Section 4.7.15, "Second internal generator", on page 258).

3. The configuration of the mixer measurement is now prepared and can be applied using the "Finish" button. If changes were made in the previous wizard steps, it is highly recommended to recalibrate the channel.



Depending on your selection, the wizard either closes without calibration or launches one of the mixer measurement-specific SMARTerCal wizards (see "SMARTerCal (Cal Unit) wizard for mixer mode" on page 330 and "SMARTerCal (Manual) wizard for mixer mode" on page 332).

Mixer Setup dialog

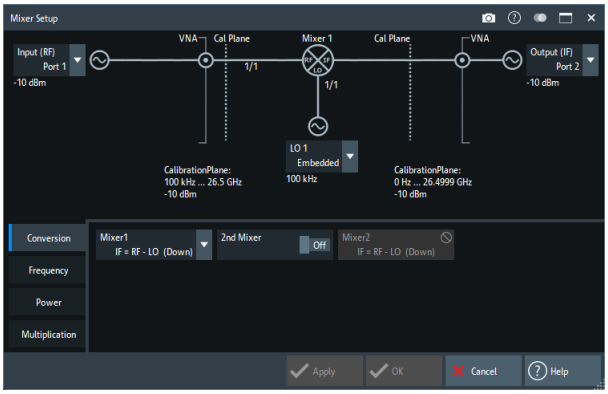
The "Mixer Setup" dialog is an example of a Multi-channel setup dialog. It lets you set up one or more channels for mixer measurements.

Access: Mixer Mode tab > "Mixer Setup"

Background information

Refer to the following sections:

- Section 4.7.3.2, "Scalar mixer measurements", on page 234
- "Two-stage mixer measurements" on page 235



The upper part the "Define Mixer Meas" dialog shows a diagram with the RF and LO signals, the mixing product (IF), and the current frequency and power ranges for all signals. The diagram is adjusted according to the "2nd Mixer" selection; see "Two-stage mixer measurements" on page 235.

In the lower part of the dialog, the frequency and power settings are performed.

Port selection

The port selection lists in the upper part of the dialog contain all analyzer ports or external generators which can provide the RF signal and local oscillator (LO) signals. The fractional numbers in the rectangles indicate the frequency conversion settings from the "Frequency" tab. The receive port for the IF signal is selected on the right side of the diagram.

Generators must be configured explicitly in the "External Generators" dialog before they appear in the list. Use the "Generator" button in the Mixer Mode tab to open it. "None" means that the input signal at LO (if available) is not controlled by the analyzer.

Remote command:

```
[SENSe<Ch>:]FREQuency:CONVersion:MIXer:RFPort
[SENSe<Ch>:]FREQuency:CONVersion:MIXer:IFPort
[SENSe<Ch>:]FREQuency:CONVersion:MIXer:LOPort<Stg>
```

Conversion

Selects the frequency of the IF signal.

The output signal (IF signal) of each mixer can be at the sum or at the difference of the RF and LO input frequencies.

- IF = RF – LO means that the RF signal is down-converted (upper sideband). The analyzer automatically switches to IF = LO – RF if the LO frequency is above the RF input frequency.

- IF = LO – RF means that the LO signal is down-converted (lower sideband). The analyzer automatically switches to IF = RF – LO if the RF input frequency is above the LO frequency.
- IF = LO + RF means that the RF input signal is up-converted.

To measure at different mixer output frequencies (e.g. to analyze the isolation for IF = RF or higher-order mixing products), use the "Define Port Settings" dialog (see Section 5.12.2.2, "Port Settings dialog", on page 686).

The basic formulas described above are adjusted to the mixer measurement settings:

- If the RF or one of the LO signals is in Auto mode (instead of IF), the formulas above are automatically converted. Example: LO 1 in Auto mode, down-conversion (lower sideband) --> LO 1 = IF + RF.
- IF = LO – RF means that the LO signal is down-converted (lower sideband). The analyzer automatically switches to IF = RF – LO if the RF input frequency is above the LO frequency.
- If two mixer stages are present, the conversion formulas are applied twice; see Two-stage mixer measurements.

Remote command:

`[SENSe<Ch>:]FREQuency:CONVersion:MIXer:TFRequency<Stg>`

2nd Mixer ← Conversion

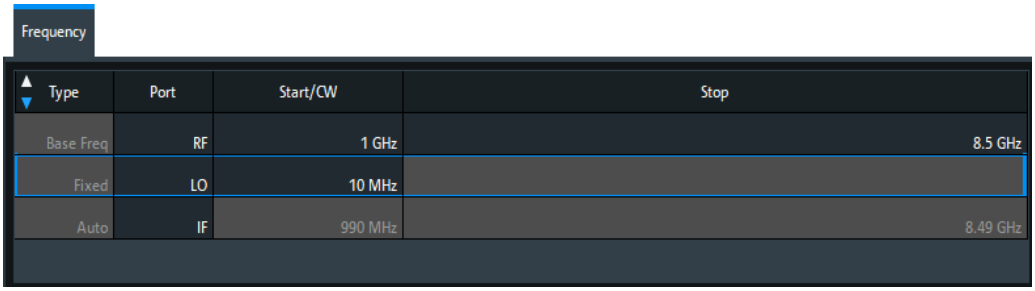
Selects the measurement for a test setup including two mixers (instead of a single one); see Two-stage mixer measurements. The signal diagram and the other control elements in the dialog are adjusted accordingly.

Remote command:

`[SENSe<Ch>:]FREQuency:CONVersion:MIXer:STAGes`

Frequency tab

The "Frequency" tab controls the frequency of the RF and LO signals and the analyzer (IF signal) frequency.



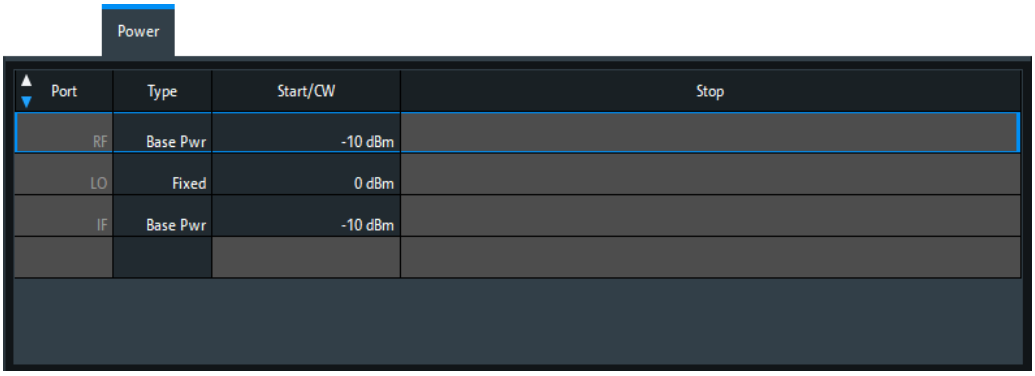
The table in the "Frequency" tab contains the following columns:

- "Type" specifies how the frequency for each signal is defined. One of the mixer input and output signals is at the analyzer's channel base frequency ("Base Freq"; the signal is swept if a frequency sweep is active; it is at fixed frequency (CW) for the other sweep types). One or two signals are at "Fixed" frequencies. The last frequency or frequency range is calculated automatically ("Auto"), depending on the other dialog settings.
- "Port" assigns the input and output ports to the predefined port frequency types. The table contains one or two LO ports, depending on whether the "2nd Mixer" is selected.

- "Start/CW" defines the start frequencies or fixed frequencies at the ports with "Base Freq" or "Fixed" frequency type and overwrites the "STIMULUS" settings. For frequency sweeps, the "Base Freq" is the frequency sweep range.
- "Stop" defines the stop value of the frequency sweep range with "Base Freq" type (for frequency sweeps only).

Remote command:
`[SENSe<Ch>:]FREQuency:CONVersion:MIXer:FUNDamental`
`[SENSe<Ch>:]FREQuency:CONVersion:MIXer:MFFixed`
`[SENSe<Ch>:]FREQuency:STARt` etc.

Power tab
The "Power" tab defines the power of the RF and the LO signals.



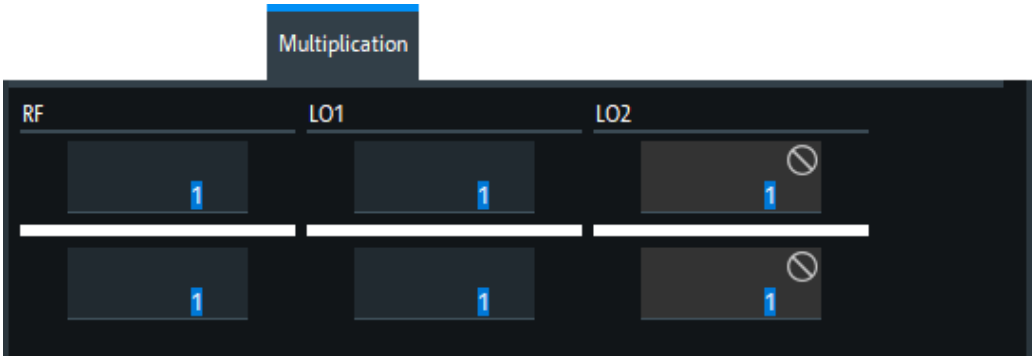
The table in the "Power" tab contains the following columns:

- "Port" contains the RF and IF ports and one or two LO ports, depending on whether the "2nd Mixer" is selected.
- "Type" specifies how the power for each signal is defined. Each of the mixer input signals RF, LO 1, and LO 2 (if present) can be either at the analyzer's channel base power ("Base Pwr"; the signal is swept if a power sweep is active; it is at fixed power (CW) for the other sweep types) or at the "Fixed" power. The same applies to the IF signal.
- "Start/CW" defines the output powers at the RF, LO 1 and LO 2 ports. The "Base Pwr" setting overwrites the power settings in the "Stimulus" softtool: For power sweeps, "Base Pwr" is the power sweep range. For the other sweep types, it is the fixed "Power" value.
  The "Fixed" power settings also appear in the "Port Configuration" dialog. The "Start/CW" power for the IF port sets the IF source power for the reverse sweep. This setting is only relevant if Load Match Correction is active.
- "Stop" shows the stop value of the power sweep range (for power sweeps only).

If a segmented frequency sweep with segment-specific powers is active, the "Base Pwr" is no longer editable but set to the minimum and maximum power of the sweep segments.

Remote command:
`SOURce<Ch>:FREQuency:CONVersion:MIXer:PMODe`
`SOURce<Ch>:FREQuency:CONVersion:MIXer:PMFixed`

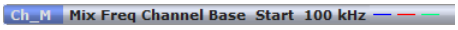
Multiplication tab
Selects the frequency conversion factors for the RF and LO 1 /LO 2 signals.



Each conversion factor is a ratio of two integer numbers. The frequency-converting device is considered to be part of the mixer system under test (MUT). In the default configuration, where the RF signal is swept and the LO signals are at fixed frequency, the conversion factors do not modify the analyzer's source signals (RF, LO 1, LO 2). They are used for the calculation of the IF frequency only.

Remote command:
`[SENSe<Ch>:]FREQuency:CONVersion:MIXer:RFMultiplier`
`[SENSe<Ch>:]FREQuency:CONVersion:MIXer:LOMultiplier<Stg>`

OK
Activates the scalar mixer mode and closes the dialog. A "Mix" label in the channel list indicates that a mixer measurement is active.



Reset Freq Conv disables the mixer measurement.

Remote command:
`[SENSe<Ch>:]FREQuency:CONVersion`

SMARTerCal (Cal Unit) wizard for mixer mode

Allows to perfom a variant of an automatic "SMARTerCal" that is specially tailored to the "Mixer Mode".

Access: Channel – [Channel Config] > "Mixer Mode" > "SMARTerCal" – "Start (Cal Unit)"

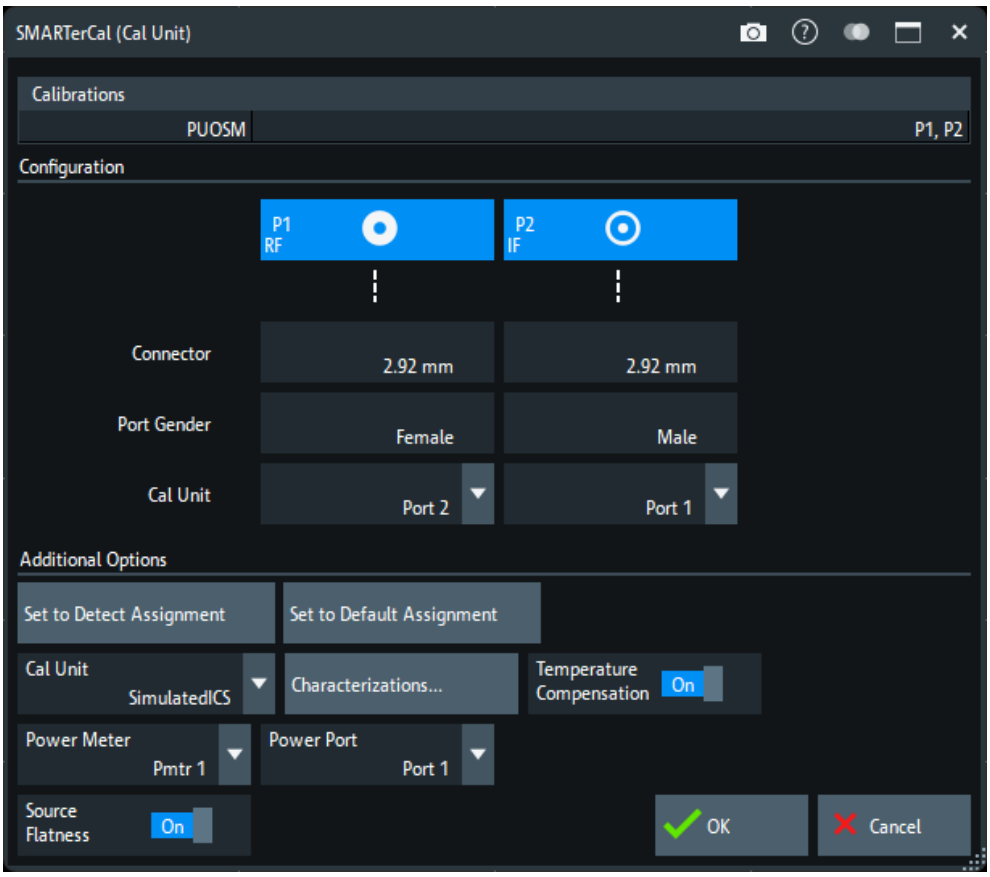
Offers similar functionality as the general SMARTerCal wizard (Cal Unit), with the following differences:

- Always all channels are calibrated
- The affected test ports are already preselected and labeled according to their role in the mixer measurement
- The adequate calibration type is already preselected
- A source flatness calibration can be automatically included as a the final calibration step

Background information
Refer to Section 4.5.7, "SMARTerCal", on page 193 for background information

The wizard proceeds as follows:

1. Specify the remaining (non-fixed) properties of the automatic SMARTerCal (required hardware, ports, connections etc.)



If the R&S ZNB is used as LO and only a 2-port Cal Unit is available, Multiple port assignments are required.

2. Perform a SMARTerCal:

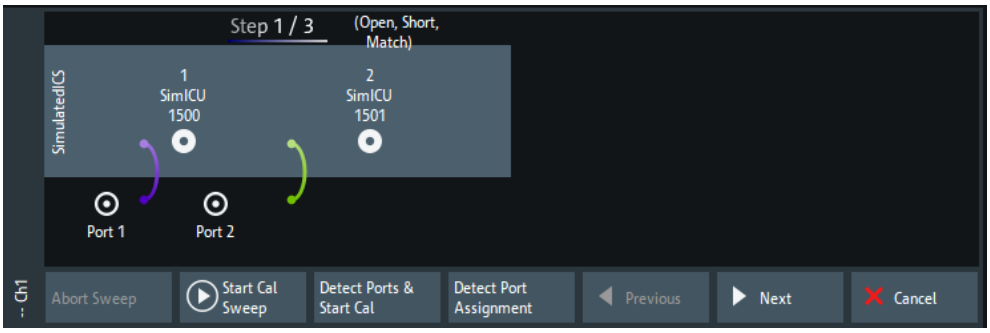


Figure 5-3: System Error Correction

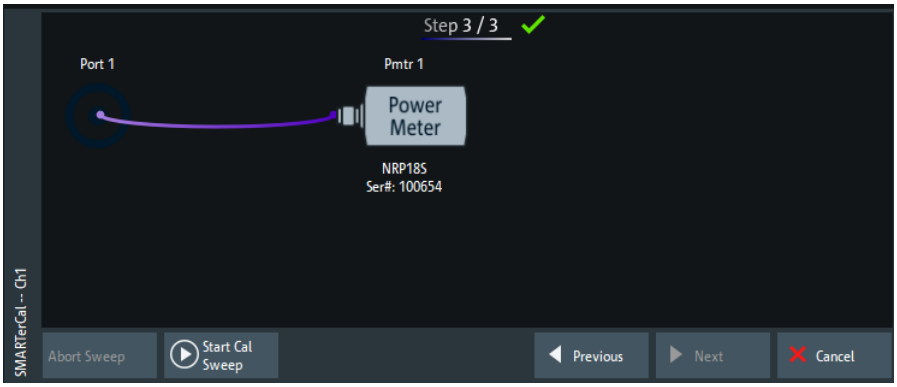


Figure 5-4: Scalar Power Calibration

SMARTerCal (Manual) wizard for mixer mode

Allows to perfom a variant of a manual "SMARTerCal" that is specially tailored to the "Mixer Mode".

Access: Channel – [Channel Config] > "Mixer Mode" > "SMARTerCal" – "Start (Manual)"

Offers similar functionality as the general SMARTerCal wizard (manual), with the following differences:

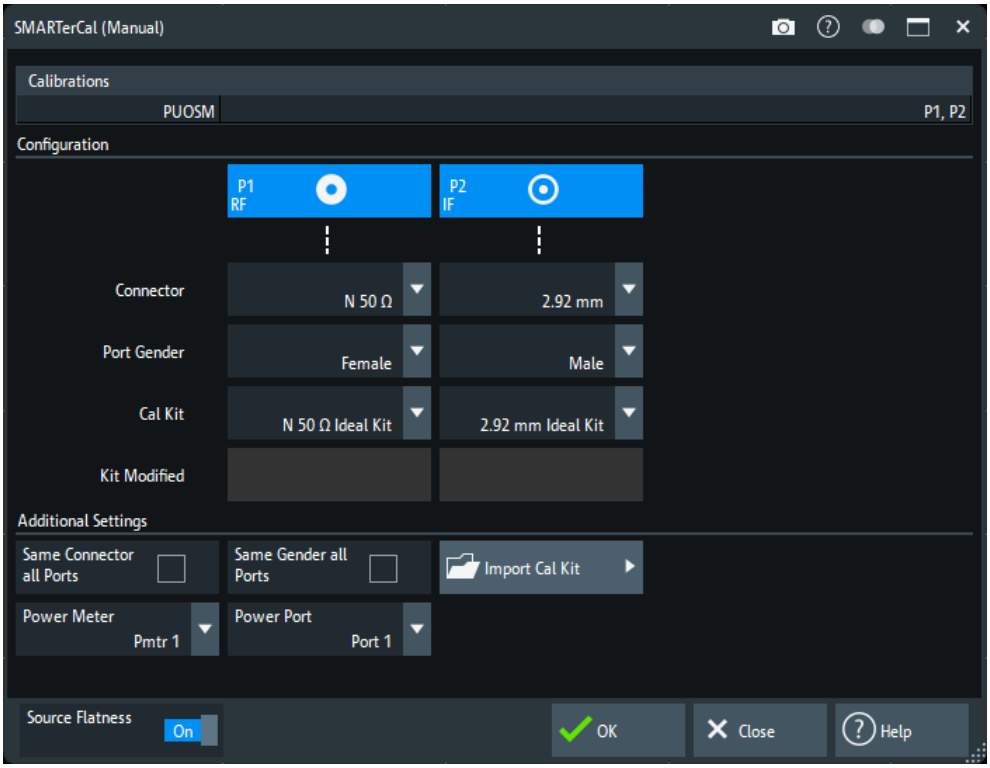
- Always all channels are calibrated
- The affected test ports are already preselected and labeled according to their role in the mixer measurement
- The adequate calibration type is already preselected
- A source flatness calibration can be automatically included as a the final calibration step

Background information

Refer to Section 4.5.7, "SMARTerCal", on page 193 for background information

The wizard proceeds as follows:

1. Specify the remaining (non-fixed) properties of the manual SMARTerCal (required hardware, ports, connections etc.)



2. Perform a SMARTerCal:



Figure 5-5: System Error Correction



Figure 5-6: Scalar Power Calibration

5.2.3 Intermodulation

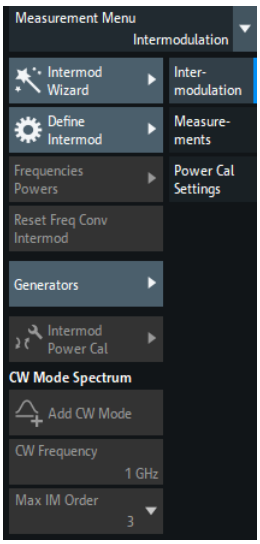
Allows you to set up an intermodulation measurement, power-calibrate it and measure related properties.

Intermodulation measurements require options R&S ZNB-K4 and R&S ZNB-K14. If these options are not available, the "Measurement Menu" item "Mixer Measurements" and the corresponding tabs are not available.

Access: Trace – [Meas] > "Measurement Menu: Intermodulation"

Background information

Refer to Section 4.7.3.3, "Intermodulation measurements", on page 238.



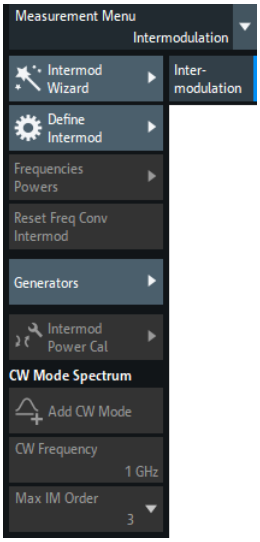
5.2.3.1 Intermodulation tab

Allows you to set up the channel for intermodulation measurements.

The Intermodulation Presetting wizard guides through the measurement configuration. The measured parameters can be selected on the Measurements tab.

Controls on the Intermodulation tab

If external switch matrices are used, intermodulation measurements are not supported. Hence, except for the "Generators..." button the controls on this tab are disabled.



The buttons on the "Intermodulation" tab open the following dialogs:

- **"Intermod Wizard":** See "Intermodulation Presetting wizard" on page 336
- **"Define Intermod":** See "Define Intermodulation Distortion Measurement dialog" on page 338
- **"Frequencies Powers":** See "Frequencies and Power dialog" on page 341
- **"Generators".:** See Section 5.17.5.2, "External Generators dialog", on page 851
- **"Intermod Pwr Cal":** See "Intermod. Pwr. Cal wizard" on page 342

Reset Freq Conv Intermod
Disables the intermodulation measurement mode for the active channel and switches back to normal mode (no frequency conversion). The intermodulation settings are maintained, however, the intermodulation traces are eliminated, and the default trace (S_{21}) is displayed.

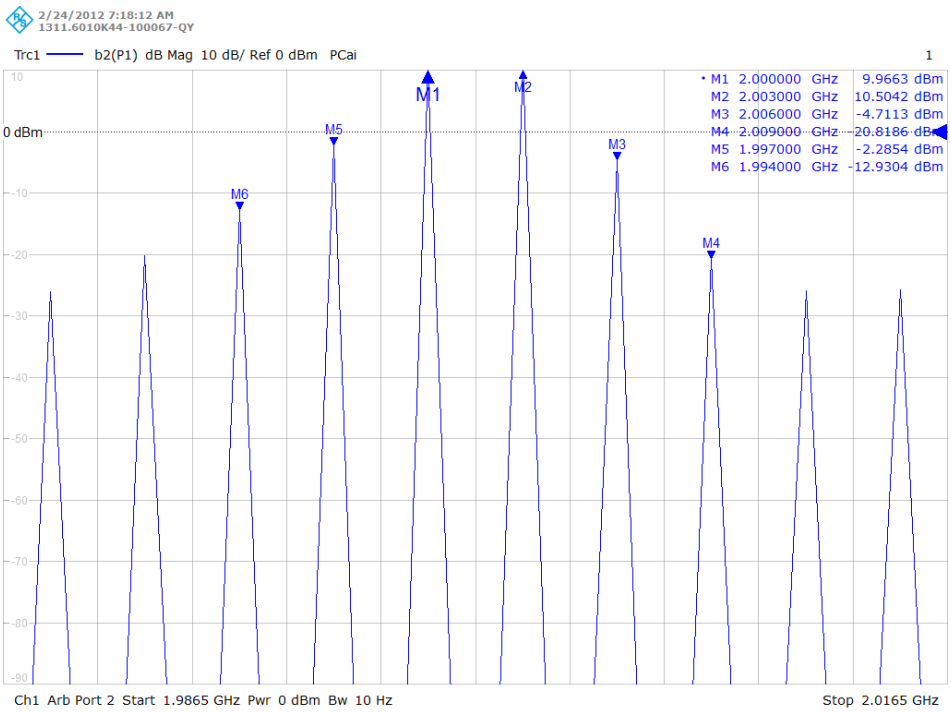
Remote command:
`[SENSe<Ch>:]FREQuency:IMODulation:CONVersion`

CW Mode Spectrum
In CW spectrum mode, the intermodulation measurement is performed at fixed frequency of the lower tone ("CW Frequency") and the upper tone ("CW Frequency" + "Tone Distance"). The analyzer displays intermodulation products up to a selectable order ("IM Order") around the lower and upper tone frequencies. The channel settings differ from the swept intermodulation measurement, therefore a new channel is created when the spectrum measurement is activated ("Add CW Mode").

The following example shows an intermodulation spectrum with the following settings:

- CW Frequency: 2 GHz (lower tone, position of marker M1)
- Tone Distance: 3 MHz (defines the upper tone, position of marker M2)
- Max. IM Order: 9.
  From left to right, the peaks correspond to the following intermodulation products:
  – Lower IM9
  – Lower IM7
  – Lower IM5 (marker M6)

– Lower IM3 (marker M5)
– Lower tone (marker M1)
– Upper tone (marker M2)
– Upper IM3 (marker M3)
– Upper IM5 (marker M4)
– Upper IM7
– Upper IM9



- "Add CW Mode"
  Activates a new channel for the spectrum measurement. The "CW Frequency" plus half the "Tone Distance" defines the center of the diagram.
- "Max IM Order"
  Defines the width of the spectrum measurement.

Remote command:

```
[SENSe<Ch>:]FREQuency[:CW]
[SENSe<Ch>:]FREQuency:IMODulation:SPECtrum:MORDer
[SENSe<Ch>:]FREQuency:IMODulation:SPECtrum[:STATe]
```

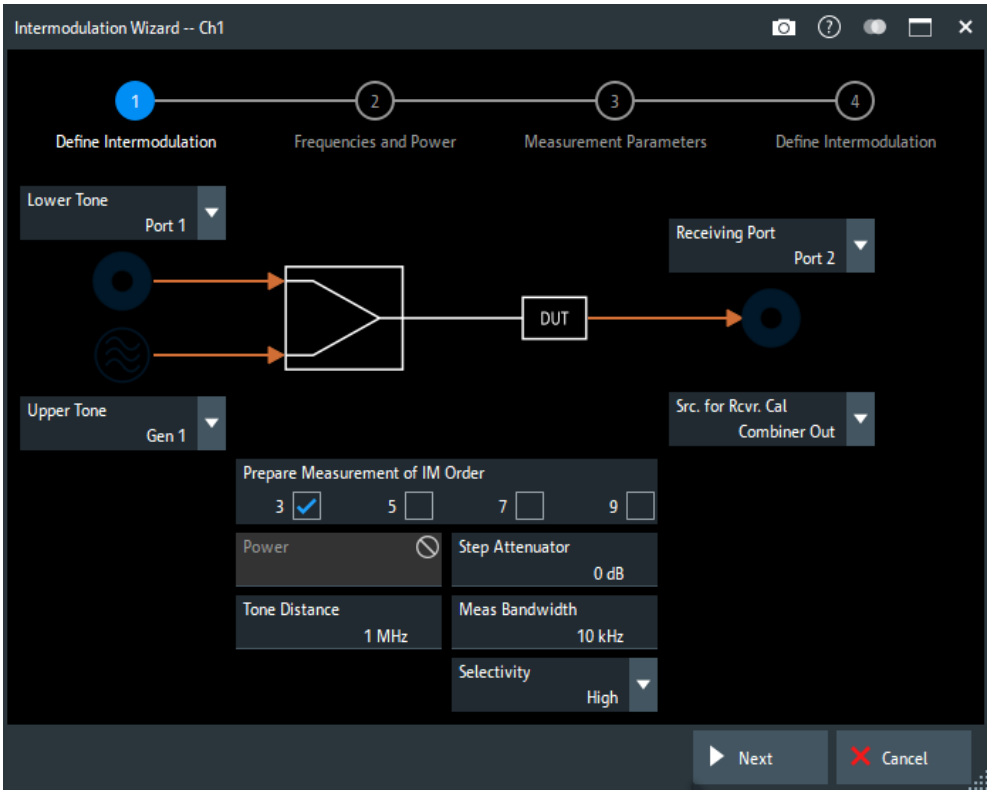
Intermodulation Presetting wizard

The "Intermodulation Presetting" wizard guides the user through the setup of an intermodulation measurement.

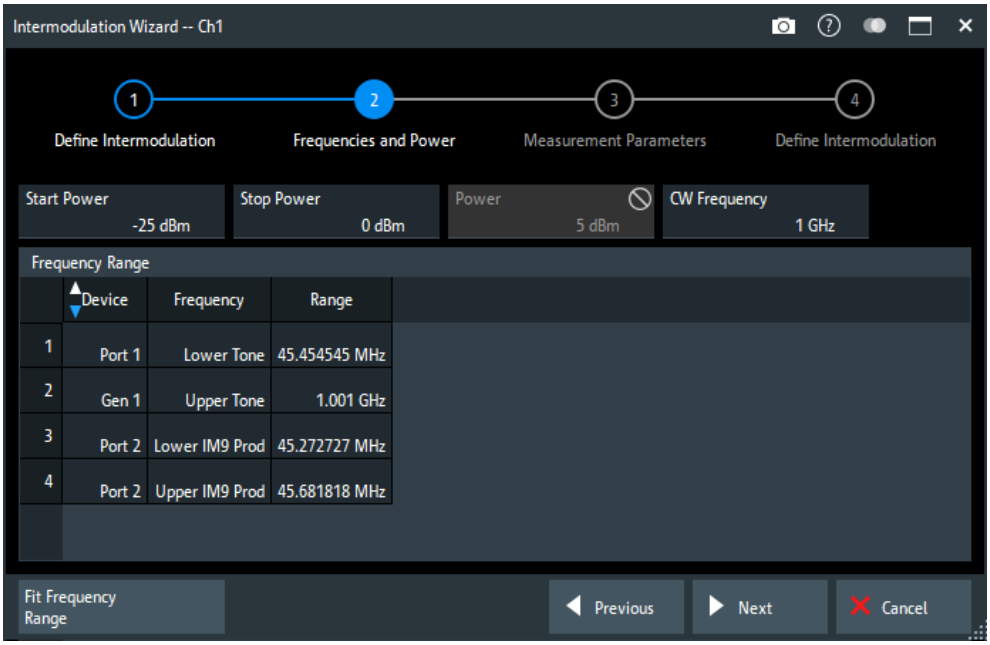
Access: Channel – [Channel Config] > "Intermodulation" > "Intermod Wizard"

The wizard proceeds in 4 steps:

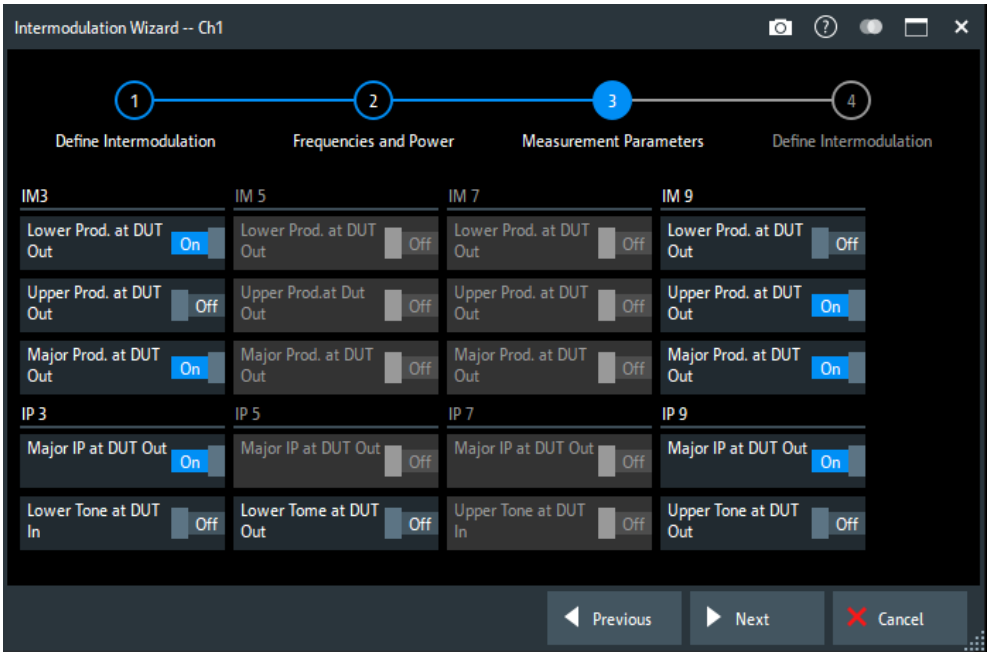
1. Configure intermodulation measurement. Refer to Section 5.2.3.1, "Intermodulation tab", on page 334.



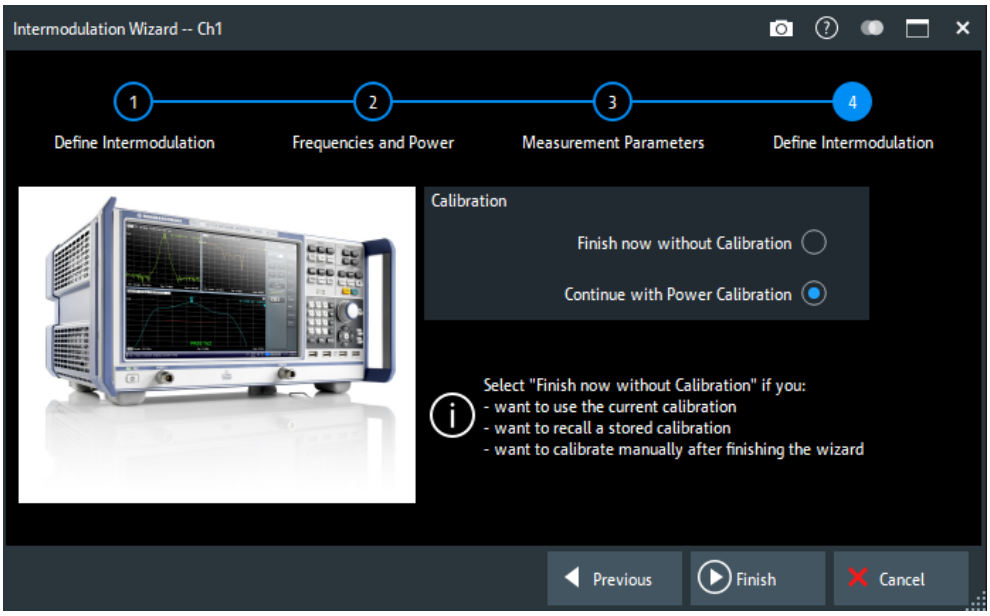
2. Define the (frequency or power) sweep ranges for the lower tone. Refer to "Frequencies and Power dialog" on page 341.



3. Select the measurement results.
Refer to Section 5.2.3.2, "Measurements tab", on page 344.



4. Finish the wizard with or without power calibration. Refer to "Intermod. Pwr. Cal wizard" on page 342.

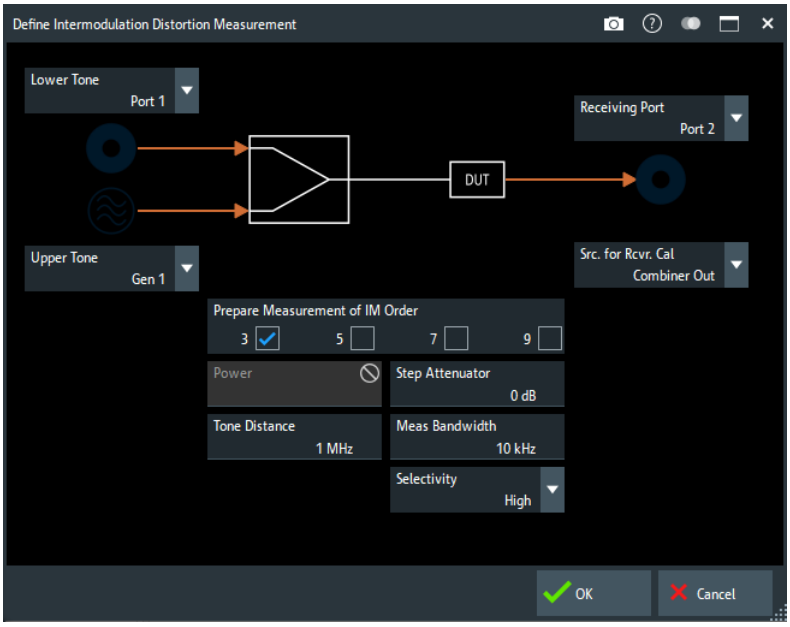


Define Intermodulation Distortion Measurement dialog

The dialog configures the intermodulation measurement (except for power and frequency settings). The "OK" button is available when a valid configuration has been defined.

Access: Channel – [Channel Config] > "Intermodulation" > "Define Intermod"

Independent sources for the lower and upper tone requires a R&S ZNB with two internal sources (see Section 4.7.15, "Second internal generator", on page 258). Alternatively an external generator can be used, see Section 4.7.3.3, "Intermodulation measurements", on page 238.



Lower Tone
Selects an analyzer port as a source of the lower tone signal.

Remote command:
`[SENSe<Ch>:]FREQuency:IMODulation:LTONe`

Upper Tone
Selects an analyzer port or external generator as a source of the upper tone signal. The source must be different from the lower tone source, i.e. you cannot combine coupled generator ports for the lower and upper tones. On a four-port R&S ZNB with Second internal generator, for example, you cannot combine ports 1 and 2 or ports 3 and 4.

If no internal second source is available, the upper tone must be provided by an external generator.

Remote command:
`[SENSe<Ch>:]FREQuency:IMODulation:UTONe`

Receiving Port
Selects the receiving port for the signal from the output of the DUT.

Remote command:
`[SENSe<Ch>:]FREQuency:IMODulation:RECeiver`

Src. for Rcvr. Cal
Selects the signal source for the power calibration of the receiving port.

In general, the combiner output signal is most convenient and provides the simplest calibration procedure. However, it is also possible to select a free analyzer or generator port, with the downside of an additional source power calibration step.

Selecting a different source is necessary, if the bandwidth of the "Combiner Out" signal does not fully cover the required receiver frequency range. For example, if a filter is used at the combiner output to cut off the intermodulation products originating from the combiner.

Remote command:
`SOURce<Ch>:POWer:CORRection:IMODulation:PORT`

Power
Power of the lower and of the upper tone. Both powers are always kept equal. For power sweeps, the start and stop power must be set in the Frequencies and Power dialog.

Remote command:
`SOURce<Ch>:POWer<PhyPt>[:LEVel][:IMMediate][:AMPLitude]`

Tone Distance
Defines the frequency difference between the upper and the lower tone. The tone distance is kept constant across the entire sweep.

Remote command:
`[SENSe<Ch>:]FREQuency:IMODulation:TDIStance`

Prepare Measurement of IM Order
Selects the orders of the measured intermodulation products. Each order requires an additional measurement at a different frequency, therefore the measurement time increases with the number of selected orders. If a higher-order IM is selected, its frequency can exceed the upper or lower frequency limit of the analyzer. Use the "Fit Frequency Range" button in the Frequencies and Power dialog to restrict the sweep range accordingly.

Remote command:
`[SENSe<Ch>:]FREQuency:IMODulation:ORDer<Im>[:STATe]`

Step Attenuator
Defines the value for the receiver step attenuator at the receiving port. Same setting as in the "Receiver Power" tab of the "Power Bw Avg" softtool; see "Step Attenuators" on page 532.

Remote command:
`[SENSe<Ch>:]POWer:ATTenuation`

Meas Bandwidth
Defines the bandwidth of the IF filter at the receiving port.

The setting is identical with the "Bandwidth" setting on the "Bandwidth" tab of the "Power Bw Avg" softtool; see Section 5.9.3, "Bandwidth tab", on page 533.

Remote command:
`[SENSe<Ch>:]BWIDth[:RESolution]`

Selectivity
Defines the selectivity of the IF filter of the receiving port.

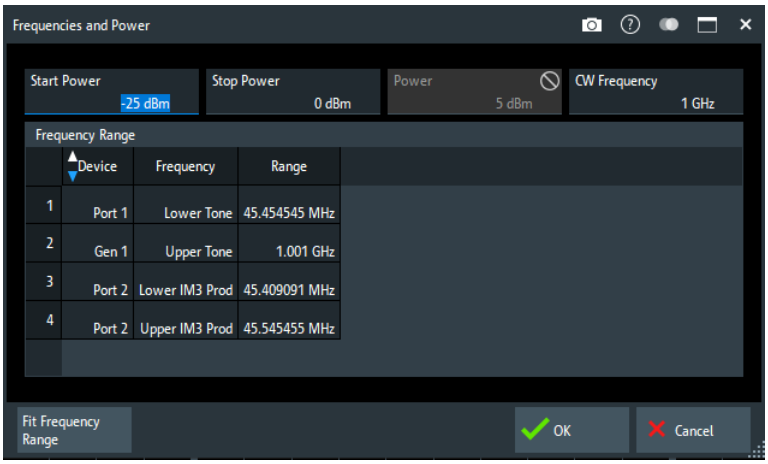
The setting is identical with the "Selectivity" setting on the "Bandwidth" tab of the "Power Bw Avg" softtool; see Section 5.9.3, "Bandwidth tab", on page 533.

Remote command:
`[SENSe<Ch>:]BWIDth[:RESolution]:SELect`

Frequencies and Power dialog

The "Frequencies and Power" dialog defines the (frequency or power) sweep ranges for the lower tone. The frequency of the upper tone is equal to the lower tone frequency plus the "Tone Distance" defined in the "Define Intermodulation Distortion Measurement" dialog. Its power is equal to the lower tone power.

Access: Channel – [Channel Config] > "Intermodulation" > "Frequencies Powers"



Start Frequency / Stop Frequency / Power CW Frequency
The sweep range settings depend on the active sweep type (frequency or power sweep). The frequency and power settings for the lower tone are identical with the "Stimulus" settings, see Section 5.8, "Stimulus softtool", on page 526.

Remote command:
`[SENSe<Ch>:]FREQuency:STARt`
`[SENSe<Ch>:]FREQuency:STOP`
`[SENSe<Ch>:]FREQuency[:CW]`
`SOURce<Ch>:POWer<PhyPt>:STARt`
`SOURce<Ch>:POWer<PhyPt>:STOP`
`SOURce<Ch>:POWer<PhyPt>[:LEVel][:IMMediate][:AMPLitude]`

Frequency Range
The "Frequency Range" table shows the frequency ranges for the lower and upper tone and for the highest-order measured IM product, depending on the current frequency settings. For example, the lower IM3 product is measured at the lower tone frequency minus the tone difference. A red background in the "Range" column indicates that the frequency range exceeds the upper or lower frequency limit of the analyzer.

Fit Frequency Range
If one of the ranges exceeds the analyzer limits, the "Fit Frequency Range" button restricts the lower tone sweep range so that the analyzer can measure all selected IM products.

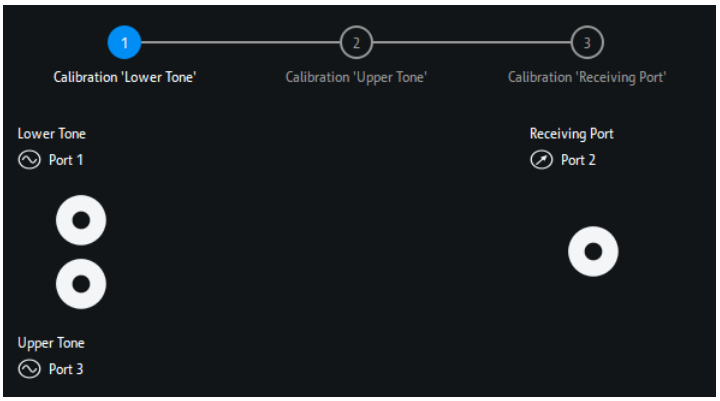
Intermod. Pwr. Cal wizard

The "Intermod. Pwr. Cal" wizard guides you through the power calibration for the intermodulation measurement. A power calibration ensures:

- accurate levels at the input of the DUT (source power calibration for the lower and upper tone)
- accurate power measurement of the intermodulation quantities (receiver calibration)

The necessary calibration steps are automatically performed across the entire frequency or power range of the intermodulation measurement.

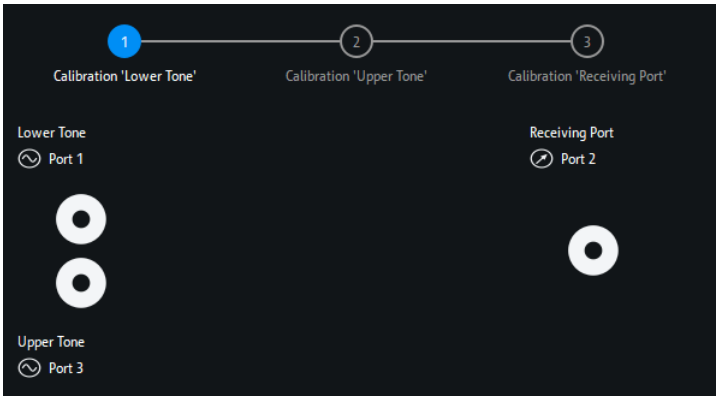
Access: Channel – [Channel Config] > "Intermodulation" > "Intermod Power Cal"



Start Cal Sweep
Starts the intermodulation power calibration. An external combiner and a power meter are required for the calibration setup (see Section 5.17.4.2, "External Power Meters dialog", on page 845). The lower and upper tone sources are combined via the external combiner.

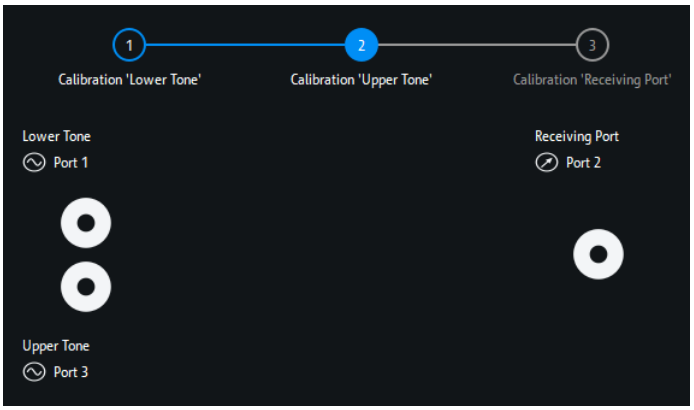
The calibration is performed in three steps:

- Source power calibration for the lower tone signal. The external power meter is connected to the combiner output. It measures the source power at the calibration plane (input of the DUT). During the lower tone source power calibration, the upper tone is switched off.



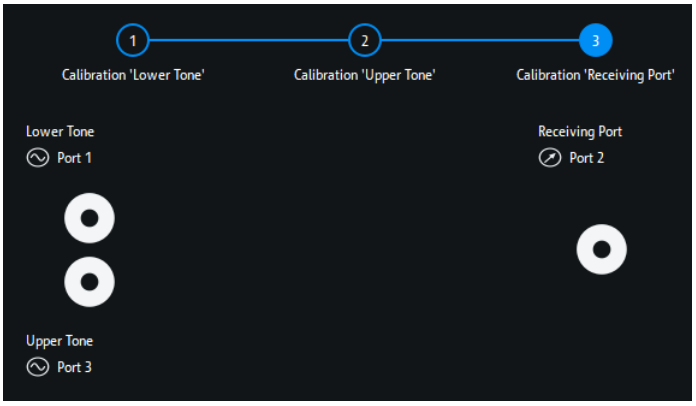
If a frequency conversion measurement is active, a second source power calibration is performed at the converted lower tone frequency (receiver frequency). This second source power calibration is the basis for the receiver calibration in step 3.

- Source power calibration for the upper tone signal. The setup of step 1 is maintained. During the upper tone source power calibration, the lower tone is switched off.



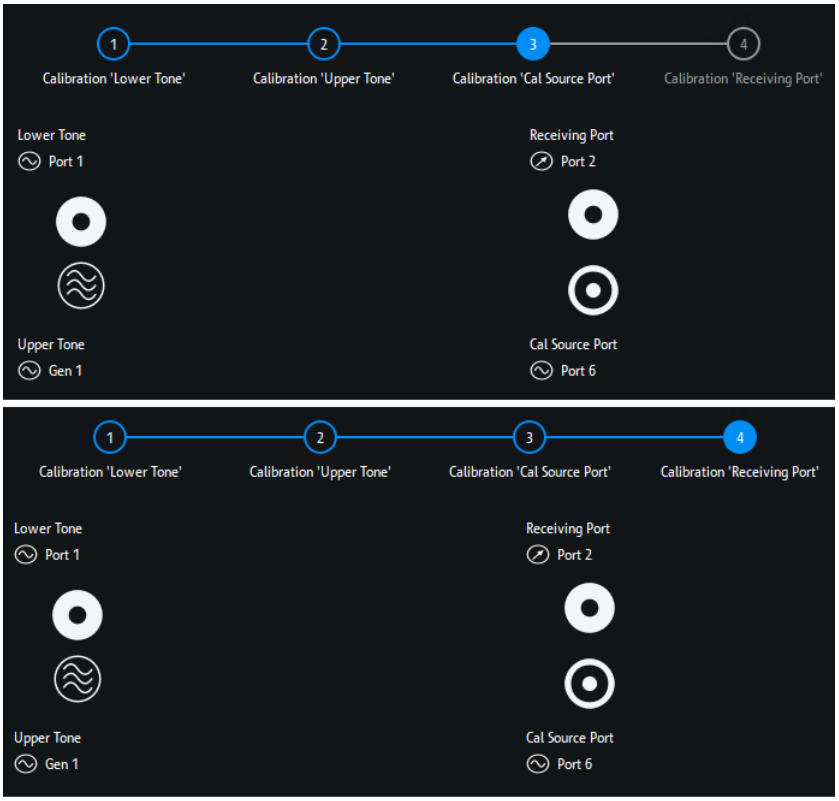
For frequency conversion measurements, no new source power calibration at the receiver frequency is performed.

- Receiver power calibration for the measured intermodulation products. The combiner output is connected to the receiving port. The receiving port is calibrated at the frequency of the intermodulation products using the signal from the lower tone signal calibrated in the first step.



If a frequency conversion measurement is active, a second receiver power calibration is performed at the converted lower tone frequency.

If a free analyzer or generator port is used for the receiver power calibration (see "Src. for Rcvr. Cal" on page 339), an additional source power calibration for this port is required. The additional calibration step is performed before step 3; the external power meter must be connected to the calibrated port.



The progress of the calibration is monitored in the calibration sweep diagram, in close analogy to an ordinary power sweep; see "Calibration Sweep Diagram" on page 596.

Remote command:

```
SOURce<Ch>:POWer<PhyPt>:CORRection:IMODulation:LTONe[:ACQuire]
SOURce<Ch>:POWer<PhyPt>:CORRection:IMODulation:UTONe[:ACQuire]
SOURce<Ch>:POWer:CORRection[:ACQuire]
[SENSe<Ch>:]CORRection:POWer<PhyPt>:IMODulation:ACQuire
```

Apply

After all calibration steps have finished successfully, "Apply" activates the resulting power calibrations and closes the "Intermod. Pwr. Cal" wizard.

5.2.3.2 Measurements tab

Selects intermodulation products and derived results as measured quantities. The intermodulation products occur at frequencies which correspond to sums and differences of the upper and lower tone frequencies and their integer multiples.

The intermodulation measurement requires options R&S ZNB-K4 and R&S ZNB-K14. The results are available when the measurement has been properly configured on the Intermodulation tab.

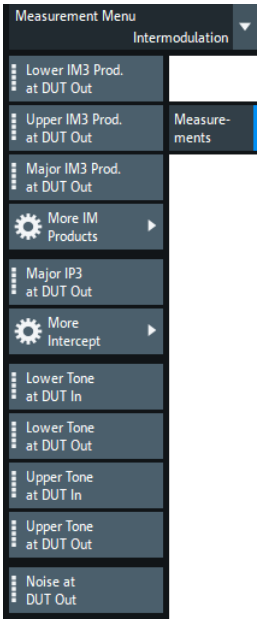
Background information

Refer to the following sections:

- Section 4.7.3.3, "Intermodulation measurements", on page 238
- "Intermodulation quantities" on page 242

The labels for the intermodulation quantities in the trace list are identical to the parameters used for remote control; see `CALCulate<Ch>:PARameter:SDEFine`.

Controls on the Measurements tab



The following buttons in the "Measurements" tab open associated dialogs:

- "More IM Products": see "IM Products dialog" on page 346
- "More Intercept": see "Intercept Points dialog" on page 347

... at DUT Out/In

Note: The measurement of higher-order intermodulation products must be enabled explicitly in the "Define Intermodulation" dialog, see "Define Intermodulation Distortion Measurement dialog" on page 338.

- The "Lower IM3 Product at DUT Out" is the third-order lower intermodulation product at the output of the DUT. This quantity corresponds to the outgoing wave quantity (b wave) at the output of the DUT, measured at the frequency of the lower tone minus the tone distance.
  The lower and upper IM<n> frequencies are also displayed in the "Frequencies and Powers" dialog, see "Frequencies and Power dialog" on page 341.

- The "Upper IM3 Product at DUT Out" is the third-order upper intermodulation product at the output of the DUT. This quantity corresponds to the outgoing wave quantity (b wave) at the output of the DUT, measured at the frequency of the upper tone plus the tone distance.
- The "Major IM3 Product at DUT Out" is the lower or upper IM3 product at DUT out, whichever is larger. The "Major IM3 Product" reveals the worst-case performance of the DUT.
- "Major IP3 at DUT Out" is the third-order major intercept point at the output of the DUT. This quantity corresponds to the (fictitious) "Lower Tone at DUT Out", where the major third-order intermodulation suppression (i.e. the ratio "Major IM3 Prod. at DUT Out"/"Lower Tone at DUT Out") reaches 0 dB.
- "Lower Tone at DUT In", "Lower Tone at DUT Out", "Upper Tone at DUT In", and "Upper Tone at DUT Out" are measurements of the two fundamental waves of the intermodulation measurement.
- "Noise at DUT Out" is a measure for the noise level and therefore the dynamic range of the intermodulation distortion measurement. The noise level is measured at the frequency of the lower tone minus half the tone distance.

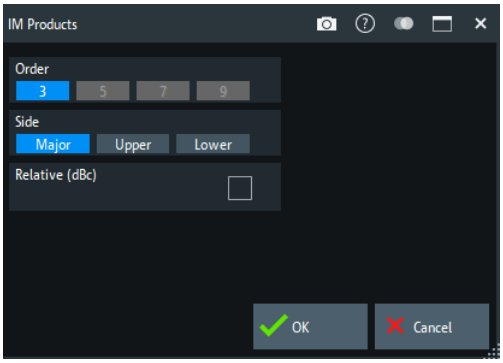
Remote command:
`CALCulate<Ch>:PARameter:MEASure`
`CALCulate<Ch>:PARameter:SDEFine`

IM Products dialog

This dialog allows you to select higher-order intermodulation products.

Access: Trace – [Meas] > "Intermod" > "More IM Products"

Background information
Refer to "Intermodulation quantities" on page 242.



Order
Defines the order of the intermodulation product.

Note that measuring a particular order must be prepared in the intermodulation channel setup (see Section 5.2.3.1, "Intermodulation tab", on page 334).

Remote command:
`CALCulate<Ch>:PARameter:MEASure`
`CALCulate<Ch>:PARameter:SDEFine`

Side
Defines the position of the intermodulation product relative to the lower and upper tones.

"Major"	Denotes the lower or upper intermodulation product, whichever is larger. The "Major" intermodulation product reveals the worst-case performance of the DUT.
"Upper"	The "Upper" intermodulation products are measured at frequencies above the upper tone.
"Lower"	The "Lower" intermodulation products are measured at frequencies below the lower tone.

Remote command:
`CALCulate<Ch>:PARameter:MEASure`
`CALCulate<Ch>:PARameter:SDEFine`

Relative
If "Relative" is selected, the intermodulation product is displayed in dB units relative to the measured lower tone level at the DUT output ("Lower Tone at DUT Out"). The relative result is often termed "intermodulation suppression".

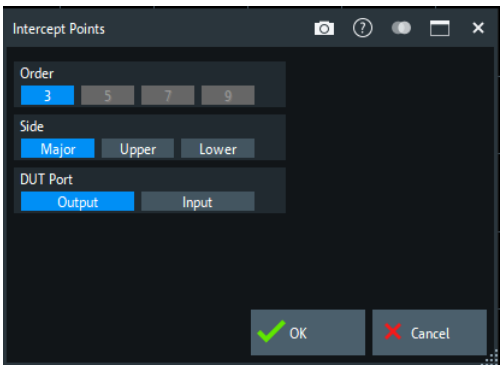
Remote command:
`CALCulate<Ch>:PARameter:MEASure`
`CALCulate<Ch>:PARameter:SDEFine`

Intercept Points dialog

This dialog allows you to select intercept points of higher order and intercept points which are measured at the input of the DUT.

Access: Trace – [Meas] > "Intermod" > "More Intercept"

Background information
Refer to "Intermodulation quantities" on page 242.



Order
Defines the order of the intercept point.

Remote command:
CALCulate<Ch>:PARameter:MEASure
CALCulate<Ch>:PARameter:SDEFine

Side
Defines the position of the intercept point relative to the lower and upper tones.

"Major" Denotes the lower or upper intercept point, whichever is **smaller**. The "Major" intercept point reveals the worst-case performance of the DUT.

"Upper" The "Upper" intercept points are measured at frequencies above the upper tone.

"Lower" The "Lower" intercept points are measured at frequencies below the lower tone.

Remote command:
CALCulate<Ch>:PARameter:MEASure
CALCulate<Ch>:PARameter:SDEFine

DUT Port
Defines if the intercept point is referenced to the DUT output or input port. Both values differ by the attenuation of the lower tone signal upon transmission through the DUT; see "Intermodulation quantities" on page 242.

Remote command:
CALCulate<Ch>:PARameter:MEASure
CALCulate<Ch>:PARameter:SDEFine

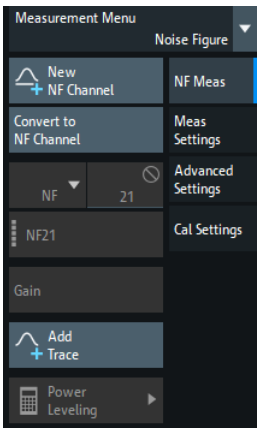
5.2.3.3 Power Cal Settings tab

See Section 5.11.3, "Power Cal Settings tab", on page 648.

5.2.4 Noise Figure

The "Noise Figure" measurement mode provides functions to set up and perform noise figure (NF) measurements with the R&S ZNB.

Access: Trace – [Meas] > "Measurement Menu: Noise Figure"



- Noise figure measurement is provided with software option R&S ZNB-K30. If this option is not installed, the "Noise Figure" tab is not available.
- Noise figure measurement is not available for frequency-converting DUTs.

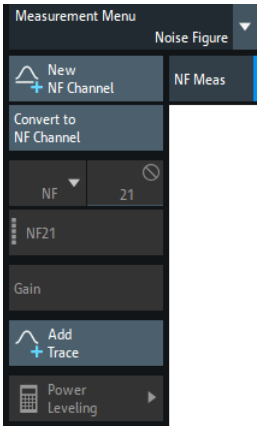
Background information

Refer to Section 4.7.6, "Noise figure measurement", on page 244.

5.2.4.1 NF Meas tab

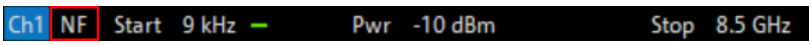
The controls on the "NF Meas" tab allow you to:

- Prepare the current channel for noise figure (NF) measurements, or create NF channels
- Define the NF and gain parameters to be measured



New NF Channel/Convert Ch<n> to NF Channel

Creates a NF channel or converts the current channel to a NF channel. The channel info changes accordingly:



Remote command:
[SENSe<Ch>:]NFIGure:DEFine

NF Parameter/NF21
The "NF Parameter" combo-boxes select the NF parameter to be measured in the current trace.

Tap or click the "NF21" button to make N_{21} the measured quantity of the active trace. Or drag the button to a diagram area to create a trace instead.

Only available if the current channel is a NF channel (see "New NF Channel/Convert Ch<n> to NF Channel" on page 349).

Remote command:
```
[SENSe<Ch>:]NFIGure:DEFine
CALCulate<Ch>:PARameter:MEASure <TraceName>, 'NF12' | 'NF13'
| ... | 'NF21' | 'NF23' | ...
CALCulate<Ch>:PARameter:SDEFine <TraceName>, 'NF12' | 'NF13'
| ... | 'NF21' | 'NF23' | ...
```

Gain
This button selects the measured gain of the DUT, as used for the noise figure calculation.

Note that in NF channels, the gain is calculated as the ratio between b-wave and a-wave, *without* mismatch correction. Therefore it is *not* identical to a vector corrected S-parameter measurement (e.g. S21).

Remote command:
```
CALCulate<Ch>:PARameter:MEASure <TraceName>, 'b1/a2(P2)' |
'b1/a3(P3)' | ... | 'b2/a1(P1)' | 'b2/a3(P3)' | ...
CALCulate<Ch>:PARameter:SDEFine <TraceName>, 'b1/a2(P2)' |
'b1/a3(P3)' | ... | 'b2/a1(P1)' | 'b2/a3(P3)' | ...
```

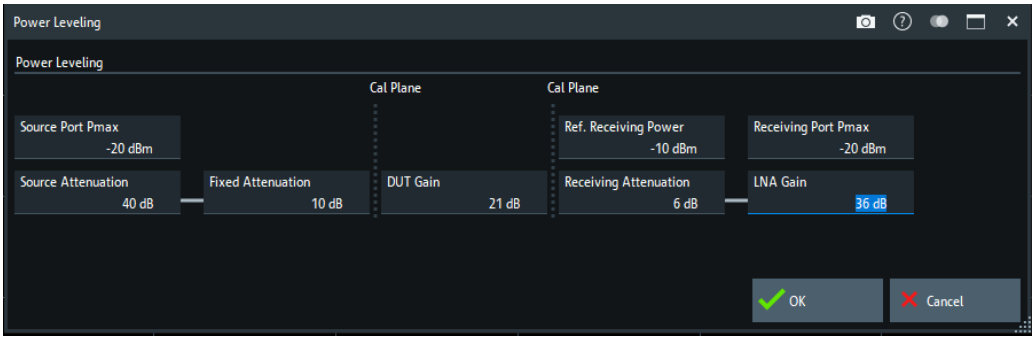
Add Trace
See "Add Trace" on page 451.

Power Leveling
Opens the Power Leveling dialog that allows you to configure the measurement environment in terms of the power levels, external gains, and external attenuations.

5.2.4.2 Power Leveling dialog

This dialog allows you to configure the measurement environment in terms of the power levels, external gains, and external attenuations for the measurement setup depicted.

Access: NF Meas tab > "Power Leveling"



Driving Port Pmax
Defines the maximum power of the driving VNA port (port 1 for NF_{21}).

Remote command:
`SOURce<Ch>:NFIGure:PLEVeling:DMAX`

Source Attenuation
Allows you to specify the attenuation you insert at the driving port. For maximum flexibility, we recommend the external step attenuator R&S RSC.

Remote command:
`SOURce<Ch>:NFIGure:PLEVeling:SATTenuation`

Fixed Attenuation
Allows you to specify the fixed attenuation you applied at the DUT input.

Remote command:
`SOURce<Ch>:NFIGure:PLEVeling:FATTenuation`

DUT Gain
Allows you to specify the (maximum) gain of your DUT.

Remote command:
`SOURce<Ch>:NFIGure:PLEVeling:DGAin`

Ref. Receiving Power
See "Ref. Receiving Power" on page 355.

Reveiving Port Pmax
Specifies the intended maximum power at the receiving VNA port (port 2 for NF_{21}).

Remote command:
`SOURce<Ch>:NFIGure:PLEVeling:RMAX`

Receiving Attenuation
Allows you to specify the fixed attenuation you applied at the DUT output.

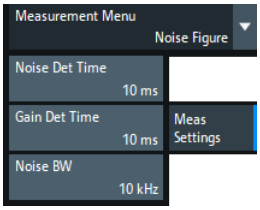
Remote command:
`SOURce<Ch>:NFIGure:PLEVeling:RATTenuation`

LNA Gain
Allows you to specify the gain of the LNA you insert at the receiving VNA port.

Remote command:
SOURce<Ch>:NFIGure:PLEVeling:LGAin

5.2.4.3 Controls on the Meas Settings tab

The "Meas Settings" tab allows you to define the detector times and IF bandwidth for noise figure measurements.



Noise Det Time
For measuring the noise power, the root-mean-square (RMS) of each measurement sample is calculated.

The noise detector measurement time sets the detector time at each measurement point for the RMS detector, allowing to choose how many RMS values are used at each frequency. A longer detector time and therefore a higher number of RMS values results in a closer estimation of the real noise power.

Remote command:
[SENSe<Ch>:]NFIGure:NTIMe

Gain Det Time
Sets the detector time at each measurement point for the average detector, which is used for the measurement of the DUT gain. Using the AVG detector allows you to remove statistical fluctuations from the measurement and therefore achieves a more accurate result.

Remote command:
[SENSe<Ch>:]NFIGure:GTIMe

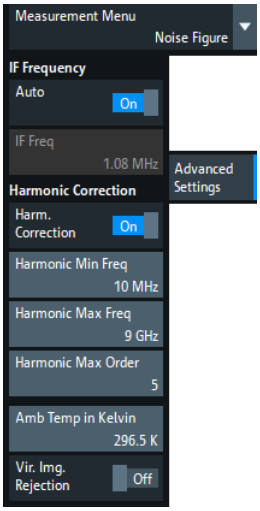
Noise BW
Sets the bandwidth used for the NF measurement and calibration. Since the receiver has no image band suppression, the noise bandwidth roughly equals two times the filter bandwidth (due to reception on both sidebands).

Important for the resulting trace noise is the product of the noise bandwidth and the chosen noise detector measurement time. Therefore, a larger noise bandwidth allows you to use a shorter measurement time or vice versa. However, due to the receiver architecture, a larger noise bandwidth also requires a higher intermediate frequency, which can influence the accuracy of measurements of narrowband DUTs.

Remote command:
See Bandwidth

5.2.4.4 Controls on the Advanced Settings tab

The "Advanced Settings" tab allows to define expert settings for NF calibration and measurement.



IF Frequency
For typical applications, the IF frequency is chosen automatically and depends on the chosen Noise BW and the measurement frequency. However, for certain measurements it can be preferable to choose a different IF frequency. The manually chosen IF frequency is applied over the full measurement frequency range.

Remote command:
`[SENSe<Ch>:]NFIGure:IFConst:STATe`
`[SENSe<Ch>:]NFIGure:IFConst`

Sideband Correction
In this section, you can compensate for the imperfect response of the VNA receivers. See "Harmonic correction" on page 247 for a detailed description. However, the overall influence on the calculated NF does not only depend on the receivers but also on the DUT. Therefore, depending on the DUT, it can be advantageous to choose a higher sideband order and a higher upper frequency limit as chosen by default.

A general recommendation is to choose a 5th order sideband correction for frequencies starting below 1 GHz. For higher frequencies, the sideband order can be reduced. For frequencies starting above 20 GHz, the effect of parasitic sidebands is typically negligible.

Example 1: A broadband amplifier with a positive gain ranging from 500 MHz to 6 GHz typically requires sideband correction during a precise NF measurement. For such an amplifier, several higher-order sidebands of the VNA receiver are within its gain bandwidth. At a measurement frequency of 500 MHz, for example, the sidebands of the VNA receiver are at 1 GHz, 1.5 GHz, 2 GHz. These bands are within the gain bandwidth of the amplifier and therefore require sideband correction. Since the DUT does not have any gain above 6 GHz, the upper frequency of the sideband correction can be set close to this value (e.g. 7 GHz).

Example 2: A narrowband amplifier with a positive gain ranging from 2.3 GHz to 2.6 GHz typically requires no sideband correction for a precise NF measurement. For such an amplifier, the sidebands of the VNA receiver (4.6 GHz to 5.2 GHz for the 2nd sideband, 6.9 GHz to 7.8 GHz for the 3rd sideband) are not within its gain bandwidth.

Note: The sideband order can be decreased after calibration. This allows you to investigate the effect of the parasitic sidebands, and to increase the measurement speed, since less additional measurements are required.

Remote command:

```
[SENSe<Ch>:]NFIGure:HARMonic[:STATe]
[SENSe<Ch>:]NFIGure:HARMonic[:MINFrequency]
[SENSe<Ch>:]NFIGure:HARMonic[:MAXFrequency]
[SENSe<Ch>:]NFIGure:HARMonic[:MAXimumorder]
```

Amb Temp in Kelvin
The ambient temperature is used as source noise temperature at the input of the DUT. The external source attenuation and additional external attenuation (cables, adapters, attenuators, etc.) are assumed to have a physical temperature equal to the ambient temperature. See "Source noise temperature" on page 246 for a detailed description.

Remote command:

```
[SENSe<Ch>:]NFIGure:TEATtenuator
```

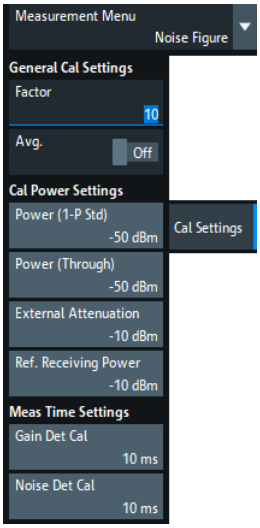
Vir. Img. Rejection
If "Virtual Image Rejection" is active, the firmware combines different measurements to remove the image band influence, which allows precise measurements of narrow spikes on "Noise Figure", "Gain" or "Noise Density" traces. By default it is disabled.

Remote command:

```
[SENSe<Ch>:]NFIGure:VIRejection[:STATe]
```

5.2.4.5 Cal Settings tab

The "Cal Settings" tab allows you to specify noise figure calibration settings such as power levels and detector times.



General Cal Settings > Factor/Avg.
Defines the averaging factor and enables/disables averaging during a noise figure calibration. See "Factor / On / Reset" on page 535.

Cal Power Settings
Power settings that are applied during a noise figure calibration.

Power (1-P Std) ← Cal Power Settings
Source power for measuring 1-port standards.

Remote command:
`SOURce<Ch>:NFIGure:CALibration:PPOWer`

Power (Through) ← Cal Power Settings
Source power for measuring Throughs.

Remote command:
`SOURce<Ch>:NFIGure:CALibration:TPOWer`

External Attenuation ← Cal Power Settings
Source power for measuring the external source attenuator (together with the Through standard).

Remote command:
`SOURce<Ch>:NFIGure:CALibration:APOWer`

Ref. Receiving Power ← Cal Power Settings
Defines the source power the R&S ZNB uses to perform the first calibration sweep of the source power calibration. See "Reference Receiver Cal Power" on page 652.

Note that for noise figure channels, the firmware disables "Use Port Power Result" to set the port power explicitly.

Remote command:
`SOURce:POWer:CORRection:PPOWer`
`SOURce:POWer:CORRection:PSELect PPOWer`

Meas Time Settings
Defines the detector times for the measurements that are performed during a noise figure calibration in channel <Ch>.

Gain Det Meas Time
Detector time of the AVG detector used for the gain measurement.

Remote command:
`[SENSe<Ch>:]NFIGure:CALibration:GTIMe`

Noise Det Meas Time
Detector time of the RMS detector used for the noise measurement.

Remote command:
`[SENSe<Ch>:]NFIGure:CALibration:NTIMe`

5.2.5 Time Domain

Offers time domain analysis of frequency domain data.

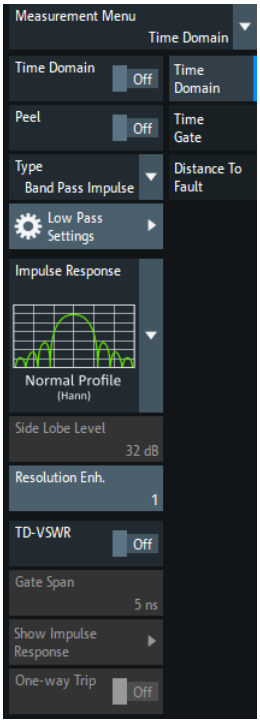
Access: Trace – [Meas] > "Measurement Menu: Time Domain"

- Time domain analysis requires option R&S ZNB-K2. If this option is not installed, "Time Domain" is not visible in the "Measurement Menu".
- Time domain analysis is only available for channels with linear or segmented frequency sweeps (see Section 5.10.2, "Sweep Type tab", on page 539). For segmented sweeps, the overall frequency grid must be harmonic.

Background information

Refer to Section 4.7.1, "Time domain analysis", on page 218.

For a comparison of the different transformation types and windows, and for application examples, please also refer to the application note 1EZ44_OE (https://www.rohde-schwarz.com/appnote/1EZ44).

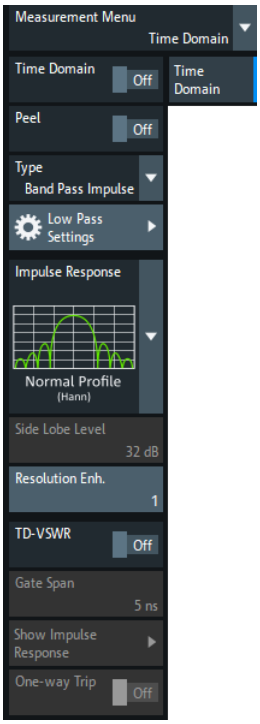


5.2.5.1 Time Domain tab

The "Time Domain" tab enables and configures the time domain representation of the measurement results.

Controls on the Time Domain tab

The contents of the "Time Domain" tab are also displayed on the "Meas" softtool for non-frequency converting DUTs.



"Low Pass Settings" opens the Low Pass Settings dialog.

Time Domain
Selects the time domain representation for the active diagram area. The softkey is enabled if a linear frequency sweep is active (see "Lin Freq" on page 540). The analyzer automatically quits time domain representation when a different sweep type is selected.

The time domain results are obtained by transforming the measured frequency sweep data into the time domain using an appropriate mathematical transformation type and frequency window ("Impulse Response"). The sweep range and the output power for the active channel is still displayed below the diagram; the displayed time interval is shown in a second line:



Trace settings in time domain representation

While the time domain representation is active, the trace settings behave as follows:

- The "Start" and "Stop" settings in the "Time Gate" tab configure the time axis.
- All trace formats including the circular diagrams are available.
- Limit lines can be defined like the limit lines for time sweeps.
- The bandfilter search functions are available for the transformed trace.
- If marker coupling is active, then the markers in the time domain and in the frequency domain are coupled with each other.

The analyzer places no restriction on the measured quantities to be transformed into the time domain. Impedances and admittances are first converted back into the equivalent S-parameter, transformed, and restored after the transformation.

See also Section 4.7.1.1, "Chirp z-transformation", on page 219.

Remote command:
`CALCulate<Chn>:TRANsform:TIME:STATe`

Define Measurement
Opens a dialog that lets you quickly select some of the most common time domain traces.



The "Step Response"/"Impedance Profile" switch changes the meaning of the numeric port input field below it. A numeric port input must be applied using "OK", the trace buttons below the input field apply their respective traces immediately.

Peel
Available for impedance profile traces $Z \leftarrow S_{ii}$ only, for other traces the control is disabled.

If active, a peeling algorithm is applied on the TDR trace. It is helpful for DUTs with impedance discontinuities in the signal path through the DUT causing distortions in the impedance trace. The discontinuities produce signal reflections and the algorithm reduces the effects of multiple reflections from the first discontinuities (early reflections).

The used algorithm does not consider path losses in the DUT (lossless peeling), so it is applicable for DUTs with low resistive and dispersive sources.

The algorithm detects the reflection effect of the first impedance discontinuity, calculates the contributions of multiple reflections by this discontinuity and subtracts them from the signal. Then, the algorithm iteratively proceeds to the next discontinuities processing them in the same way. This iterative process "peels" away the impedance distortions from the first impedance discontinuities.

Remote command:
CALCulate<Chn>:TRANsform:PEEL:STATe

Type
Selects a bandpass or lowpass time domain transform. See Section 4.7.1.2, "Band pass and low pass mode", on page 219.

To calculate a lowpass transform, the sweep points must be on a harmonic grid. Otherwise the analyzer can only calculate an approximate result and generates a warning. "Low Pass Settings" opens a dialog that allows to establish or change a harmonic grid (not available for memory traces). See "Low Pass Settings dialog" on page 361.

Remote command:
CALCulate<Chn>:TRANsform:TIME[:TYPE]
CALCulate<Chn>:TRANsform:TIME:STIMulus

Impulse Response
Selects a window type that the R&S ZNB uses to filter the trace in the frequency domain. The drop-down list shows the impulse response of a constant trace over a finite sweep range (i.e. a rectangular function) that was filtered using the different available window types. The selected window is applied to the active trace.

See also Section 4.7.1.3, "Windows in the frequency domain", on page 220.

Note: The frequency domain window is used to filter the trace before transforming it to the time domain. An independent "Time Gate" can be used after the transformation to eliminate unwanted responses (see Section 5.2.5.2, "Time Gate tab", on page 363).

The analyzer always uses a "No Profiling (Rectangle)" window to calculate the time-gated frequency domain trace, see "Time-gated frequency domain trace" on page 224.

Note that the "Causal Rise Time" filter requires a Low Pass Step transformation with Automatic Harmonic Grid enabled.

Remote command:
CALCulate<Chn>:TRANsform:TIME:WINDow

Side Lobe Level
Defines the sidelobe suppression for an "Arbitrary Sidelobes (Dolph-Chebychev)" window. The entered value is the ratio of the power of the central lobe to the power of the first sidelobe in dB.

Remote command:
CALCulate<Chn>:TRANsform:TIME:DCHebyshev

Kaiser Smooth Factor
Defines the smoothness factor of a Kaiser window.

See Table 4-22.

Remote command:
CALCulate<Chn>:TRANsform:TIME:KAISer

Gaussian Smooth Factor
Defines the smoothness factor of a Gaussian window.

See Table 4-22.

Remote command:
CALCulate<Chn>:TRANsform:TIME:GAUSsian

Exponential Smooth Factor
Defines the smoothness factor of an exponential window.

See Table 4-22.

Remote command:
CALCulate<Chn>:TRANsform:TIME:EXPonential

Causal Rise Time
Defines the rise time for the causal rise time window.

See Table 4-22.

Remote command:
CALCulate<Chn>:TRANsform:TIME:CAUSal

Resolution Enh.
Broadens the frequency range that the analyzer considers for the time domain transform by a linear factor. A factor of 1 means that the original sweep range and the measured sweep points are used; no additional assumptions are made. With higher resolution enhancement factors, the measurement data is extrapolated using a linear prediction method. As a result, the time domain resolution can be improved.

The ideal resolution enhancement factor depends on the properties of the DUT. For distance-to-fault measurements on cables, set it to 1.

Remote command:
CALCulate<Chn>:TRANsform:TIME:RESolution:EFACtor

TD-VSWR
Enables Time domain S_{VSWR} measurements.

Remote command:
CALCulate:TDVSwr[:STATe]

Gate Span
Time domain S_{VSWR} measurements rely on a time gate that is centered at the antenna's direct response (plus ring-down time), separating the direct response from the indirect responses (reflections).

"Gate Span" is equivalent to the "Span" value on the Time Gate tab.

This button is only enabled if TD-VSWR is active.

Remote command:
CALCulate<Chn>:FILTer[:GATE]:TIME:SPAN

Show Impulse Response...
To get an impression of the required Gate Span, you can quickly create a trace displaying the impulse response of the active trace in a new diagram.

This button is only enabled if TD-VSWR is active.

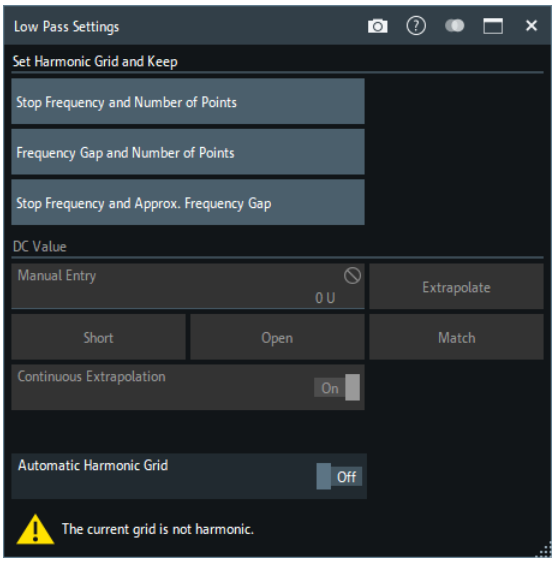
Low Pass Settings dialog

The "Low Pass Settings" dialog defines the harmonic grid for lowpass time domain transforms.

Access: Trace – [Trace Config] > "Time Domain" > "Low Pass Settings"

Background information

Refer to Section 4.7.1.4, "Harmonic grid", on page 221.



For segmented sweeps, time domain analysis requires the overall frequency grid to be harmonic. However, the "Automatic Harmonic Grid" functions for linear frequency sweeps are not available for segmented sweeps. In this case, you have to configure the harmonic grid manually (see Section 5.10.2.2, "Define Segments dialog", on page 545).

Set Harmonic Grid and Keep

The three buttons in this section provide alternative algorithms for calculating a harmonic grid, based on the current sweep points.

- Keep "Stop Frequency and Number of Points" calculates a harmonic grid based on the current "Stop Frequency" (see "Start Frequency / Stop Frequency / Center Frequency / Span Frequency" on page 527) and the current number of sweep points (see "Number of Points" on page 537). This algorithm can increase the frequency gap (i.e. the Freq Step Size).
- Keep "Frequency Gap and Number of Points" calculates a harmonic grid based on the current "Stop Frequency" and the current frequency gap (i.e. the "Freq Step Size").
- Keep "Stop Frequency and Approximate Frequency Gap" calculates a harmonic grid based on the current "Stop Frequency", increasing the "Number of Points" in such a way that the frequency gap (i.e. the "Freq Step Size") remains approximately the same. This algorithm can increase the sweep time, due to the additional sweep points introduced.

The three grids can be calculated repeatedly in any order; the analyzer always starts from the original set of sweep points.

Note: For segmented sweeps, time domain analysis requires the overall frequency grid to be harmonic. However, the "Set Harmonic Grid and Keep" function is not available and hence this control is disabled. In this case, you have to configure the harmonic grid manually (see Section 5.10.2.2, "Define Segments dialog", on page 545).

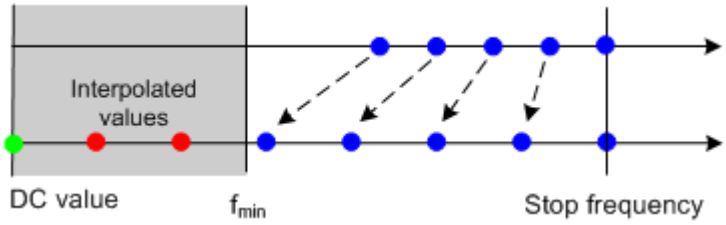
Remote command:
`CALCulate<Chn>:TRANsform:TIME:LPASs`

DC Value
The control elements in this section define the measurement result at zero frequency and in the interpolation/extrapolation range between f = 0 and f = f_{min}. They are enabled after a harmonic grid has been established.

Defining the low frequency sweep points

After calculating a harmonic grid, the analyzer must determine the value of the measured quantity at grid points below the analyzer's minimum frequency f_{min}.

The following figure shows a scenario where the harmonic grid was calculated with fixed "Stop Frequency and Number of Points". The DC value and the values at the two additional red points must be extrapolated or interpolated according to the values at the measured sweep points (blue dots).



- If the properties of the DUT at f = 0 are sufficiently well known, then it is recommendable to enter the DC value manually ("Manual Entry").
  Examples: At f = 0, the reflection factor of an open-ended cable is 1. It is –1 for a short-circuited cable and 0 for a cable with matched termination. Use the "Open", "Short" and "Match" buttons to set the corresponding value.
  In general, if a cable with known termination is measured, enter these numbers as DC values.
- The "Extrapolate" button initiates an extrapolation of the measured trace towards f = 0 and overwrites the current DC value. This function can be used for a consistency check.
- "Continuous Extrapolation" initiates an extrapolation of the measured trace towards lower frequencies, so that the missing values (green and red dots) are obtained without any additional input. The extrapolation is repeated after each sweep.

After setting or extrapolating the DC value, the analyzer then calculates the remaining values (red dots) by linear interpolation of the magnitude and phase.

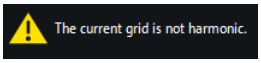
Remote command:
`CALCulate<Chn>:TRANsform:TIME:LPASs:DCSParam`
`CALCulate<Chn>:TRANsform:TIME:LPASs:DCSParam:CONTinuous`
`CALCulate<Chn>:TRANsform:TIME:LPASs:DCSParam:EXTRapolate`
`CALCulate<Chn>:TRANsform:TIME:LPFRequency`

Automatic Harmonic Grid
If enabled the frequency grid is automatically kept harmonic.

Note: For segmented sweeps, time domain analysis requires the overall frequency grid to be harmonic. However, the "Automatic Harmonic Grid" function is not available and hence this control is disabled. In this case, you have to configure the harmonic grid manually (see Section 5.10.2.2, "Define Segments dialog", on page 545).

Remote command:
`[SENSe<Ch>:]HARMonic:AUTO`

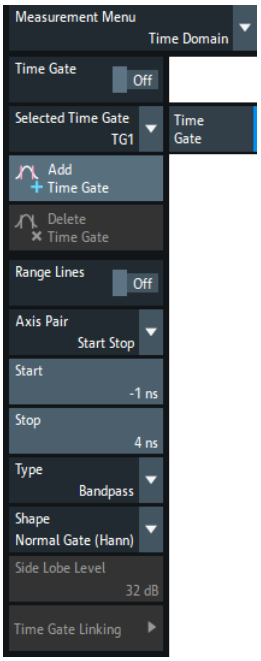
Is the Current Grid Harmonic?
The area at the bottom of the "Low Pass Settings" dialog indicates whether the current frequency grid is harmonic.



Remote command:
`[SENSe<Ch>:]HARMonic?`

5.2.5.2 Time Gate tab

Defines and activates gates in the time domain. An active time gate acts on the trace in time domain and in frequency domain representation. In time domain representation, you can use the time gate settings to eliminate unwanted responses in your signal. After switching back to the frequency domain, you will receive the frequency response of your DUT without the contribution of the unwanted responses. The time gate is independent of the frequency window used to filter the trace before transforming it to time domain.



Time Gate
Enables or disables the time gates of the active trace. "Gat" is displayed in the trace list while the time gate is active.



Remote command:
`CALCulate<Chn>:FILTer[:GATE]:TIME:STATe`

Selected Time Gate
Selects a time gate for configuration purposes.

Use Add Time Gate or Delete Time Gate to add/delete time gates to/from the active trace.

Remote command:
`CALCulate<Chn>:FILTer[:GATE]:TIME:CATalog?`
`CALCulate<Chn>:FILTer[:GATE]:TIME:SELect`

Add Time Gate
Cretates a time gate for the active trace and selects it for further configuration.

For each trace, time gates are named 'TG<n>' with positive integers <n>. Whenever you `ADD` a time gate, the highest available <n> is incremented by 1.

Remote command:
`CALCulate<Chn>:FILTer[:GATE]:TIME:ADD`

Delete Time Gate
Deletes the active trace's Selected Time Gate.

Remote command:
`CALCulate<Chn>:FILTer[:GATE]:TIME:DELete`

Axis Pair
"Start Stop" lets you define the Selected Time Gate via its "Start" and "Stop", "Center Span" lets you define it via its "Center" and "Span" value (in time).

The analyzer generates a warning if the (resulting) time span exceeds the unambiguous range which is given by $\Delta t = 1/\Delta f$, where Δf is the "Freq Step Size". Simply reduce the time span until the warning disappears.

Remote command:
`CALCulate<Chn>:FILTer[:GATE]:TIME:CENTer`
`CALCulate<Chn>:FILTer[:GATE]:TIME:SPAN`
`CALCulate<Chn>:FILTer[:GATE]:TIME:STARt`
`CALCulate<Chn>:FILTer[:GATE]:TIME:STOP`

Show Range Lines
Displays or hides red vertical lines indicating the configured time gates in a time domain diagram.

Remote command:
`CALCulate<Chn>:FILTer[:GATE]:TIME:SHOW`

Type
Defines the filter type of the selected time gate.

The filter type determines what happens to the data in the specific time region.
- A "Bandpass" filter passes all information in the specified time region and rejects everything else.
- A "Notch" filter rejects all information in the specified time region and passes everything else.

Remote command:
`CALCulate<Chn>:FILTer[:GATE]:TIME[:TYPE]`

Shape
Defines the gate shape of the selected time gate.

Filter type and gate determine how the R&S ZNB filter the trace in the time domain. The drop-down list visualizes how the time gate will affect a constant function after transformation back into the frequency domain. The selected window is applied to the active trace. The two red vertical lines represent the "Start" and "Stop" values defining the size of the time gate.

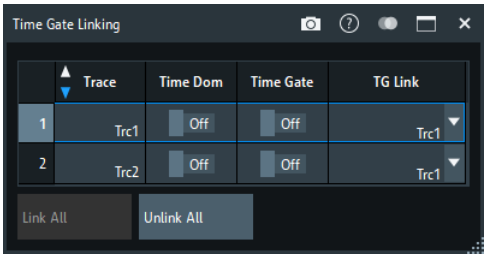
For background information, see Section 4.7.1.5, "Time gates", on page 223.

Remote command:
`CALCulate<Chn>:FILTer[:GATE]:TIME:SHAPe`
`CALCulate<Chn>:FILTer[:GATE]:TIME:WINDow`

Side Lobe Level
Defines the side lobe suppression for an "Arbitrary Gate Shape (Dolph-Chebychev)" gate. The entered value is the ratio of the power of the central lobe to the power of the first side lobe in dB.

Remote command:
`CALCulate<Chn>:FILTer[:GATE]:TIME:DCHebyshev`

Time Gate Linking
Opens a dialog that allows you to link the time gates of the traces in the active channel, i.e. to align the time gate setting across linked (~coupled) traces.



"Link All" links all traces to the last one in the table.

For your convenience, time domain representation and time gating of traces can also be enabled from this dialog.

Remote command:
`CALCulate<Chn>:FILTer[:GATE]:TIME:LINK`

5.2.5.3 Distance to Fault tab

With its optional distance-to-fault measurements, the R&S ZNB can locate faults and discontinuities on cables and transmission lines. The measured S_{ii}-parameter trace is mathematically converted to the time domain, and represented in the distance domain. The x-axis of the DtF trace (derived from S_{ii}) shows the distance from the calibrated reference plane; faults appear as peaks above a certain level.

If option R&S ZNB-K3 is not installed on the R&S ZNB, the "Distance to Fault" application is not available.

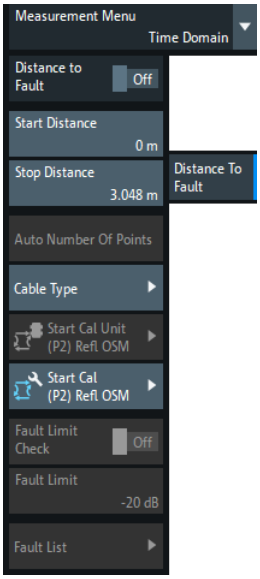
To achieve the best distance resolution, set the sweep span as large as possible and use the Auto Number of Points to adjust the number of sweep points.

Background information

Refer to Section 4.7.2, "Distance-to-fault measurements", on page 231.

Controls on the Distance to Fault tab

The controls on the "Distance to Fault" tab allow you to enable and configure a standard Distance to Fault (DtF) measurement.



A standard DtF measurement is prepared in the order from top to bottom:

1. Enable DtF measurement (see "Distance to Fault" on page 367)
2. Configure the distance window (see "Start Distance / Stop Distance" on page 367)
3. Adjust the number of sweep points and, if necessary, the frequency span (see Auto Number of Points)
4. Select – or define and select – a suitable cable type (see"Cable Type" on page 368)

5. Perform a full one-port calibration at physical port 2 (see "Start Cal Unit (P2) Refl OSM / Start Cal (P2) Refl OSM" on page 369)

   Locate the faults (peaks) by examining the trace.
   You can also let the firmware generate a list of faults by enabling Fault Limit Check and defining a suitable Fault Limit. Use Fault List to display (and export) the detected faults.

Distance to Fault
Activates/deactivates Distance to Fault representation for the active trace.

Note that "Distance to Fault" can only be enabled, if the active channel is configured to perform a linear frequency sweep.

If the active trace is a reflection trace S_{ii}, the analyzer firmware assumes that the DUT is connected to port $p=i$. Otherwise it assumes that the DUT is connected to port $p=2$.

When activating "Distance to Fault", the analyzer proceeds as follows:
- Set S_{pp} as measured quantity of the active trace
- Propose a reflection normalization at port p (see "Start Cal Unit (P2) Refl OSM / Start Cal (P2) Refl OSM" on page 369)
- Enable additional controls on the "Distance to Fault" tab
- Append the selected cable type to the trace label (see "Cable Type" on page 368)
- Display an additional "Distance Domain" trace info line below the channel line



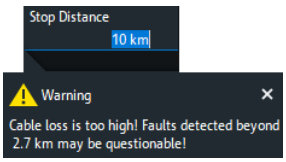
Note that all info lines (channel info, trace info,"Distance Domain" trace info) allow quick access to related parameters via specific context menus:

"Distance to Fault" can also be enabled/disabled via the context menu of the trace label.

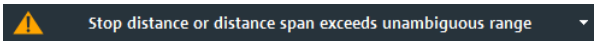
Remote command:
`CALCulate<Chn>:TRANsform:DTFault:STATe`

Start Distance / Stop Distance
Define the distance window for the Distance to Fault measurement. "Start Distance" and "Stop Distance" correspond to the left and right edge of the diagram area and are displayed in the "Distance Domain" trace info line.

Note:
- If the "Stop Distance" is so high that for the selected cable type the signal reflected at "far away discontinuities" would be too small, a warning tooltip is displayed:



- If the "Stop Distance" exceeds the unambiguous range (for the given frequency span and number of sweep points), a warning is displayed in the instrument status bar:



Use the Auto Number of Points function to adjust the number of sweep points so that the impulse response becomes unambiguous in the configured distance window.

Remote command:
```
CALCulate<Chn>:TRANsform:DTFault:STARt
CALCulate<Chn>:TRANsform:DTFault:STOP
CALCulate<Chn>:TRANsform:DTFault:CENTer
CALCulate<Chn>:TRANsform:DTFault:SPAN
```

Auto Number of Points
Selects the minimum number of sweep points that are required to provide an unambiguous display of fault locations for the active Distance to Fault trace.

If "Auto Number of Points" actually modifies the Number of Points, a tooltip is displayed:

The required number of points depends on:
- The frequency span $\Delta f = f_{stop} - f_{start}$ (see "Start Frequency / Stop Frequency / Center Frequency / Span Frequency" on page 527),
- The stop distance d_{stop} (see "Start Distance / Stop Distance" on page 367), and
- The velocity factor v of the transmission line (see "Cable Type" on page 368).

With c_0 denoting the speed of light in a vacuum, the analyzer sets the number of sweep points N to:

$$N = \max \{ \lceil 2.6 \cdot d_{stop} \cdot \Delta f / (v \cdot c_0) \rceil , 201\}$$

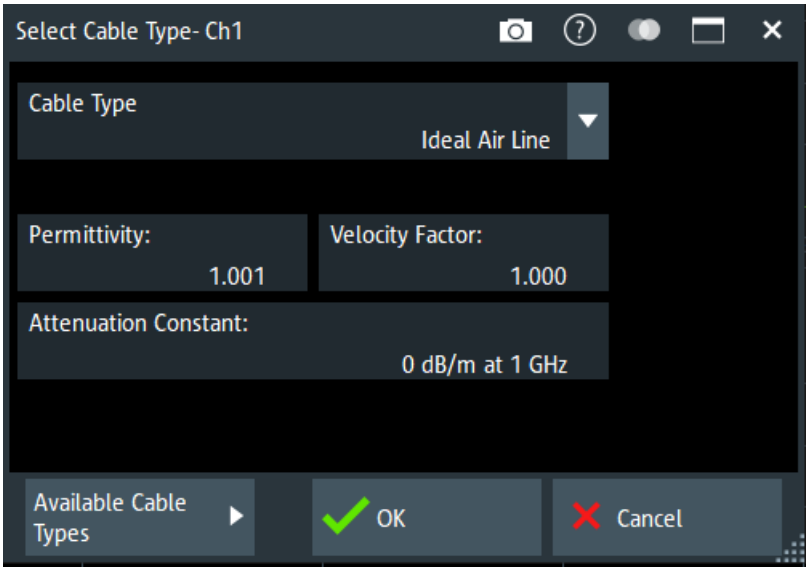
If the required number of sweep points would be higher than N_{max} = 100001 (the maximum number of sweep points of the R&S ZNB), the analyzer sets N to N_{max} and displays the following message box:

"OK" reduces the frequency span to $\Delta f = v \cdot c_0 \cdot N_{max} / (2.6 \cdot d_{stop})$, which makes N_{max} sweep points sufficient for unambiguous display (see the formula above). However, this reduction of the frequency span comes at the cost of coarser distance resolution.

"Cancel" (default) leaves the frequency unchanged.

Remote command:
```
CALCulate<Chn>:TRANsform:DTFault:POINts
```

Cable Type
Opens a dialog that allows to select a preconfigured cable type to be used for the calculation of the Distance to Fault traces on the active channel. A cable type is defined by its relative permittivity (or its velocity factor), and a frequency-dependent attenuation table.



Select "Available Cable Types" to access the list of predefined and user-defined cable types (see "Available Cable Types dialog" on page 370).

Remote command:
`CALCulate<Chn>:TRANsform:DTFault:SELect`

Start Cal Unit (P2) Refl OSM / Start Cal (P2) Refl OSM
Sets up and runs the "Calibration Unit"/"Calibration Setting" wizard to perform a full one-port calibration at physical port 2 (only port 2 enabled and calibration type "Refl OSM" preselected).

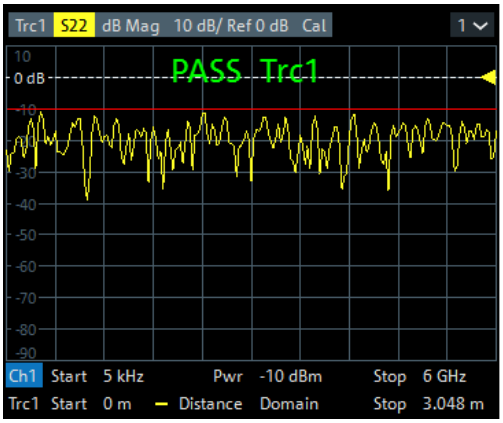
The calibration serves several purposes:
- Adjust the reference plane (distance zero).
- Normalize the trace (total reflection of the signal corresponds to a 0 dB peak).
- Avoid spurious effects, e.g. peaks on the impulse response trace that are not due to a fault.

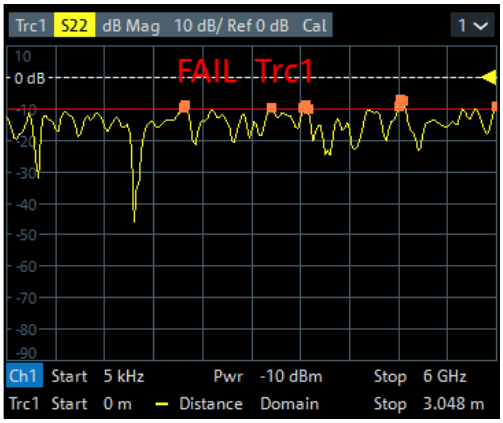
For descriptions of the calibration wizards and the corresponding remote commands, see Calibration Unit wizard and Calibration Setting wizard.

Note: Make sure to select a sufficient number of sweep points and a suitable frequency span before you start the calibration. Otherwise the measurement can yield inaccurate results.
Use Auto Number of Points to let the firmware assist you.

Fault Limit Check
Enables/disables checking the current "Distance to Fault" trace for spikes above the given Fault Limit (red line).

The fault limit is defined relative to the 0 dB-line in the test diagram, i.e. the peak response value for total reflection after proper calibration.





Note that the fault limit is handled like any other limit line. When "Fault Limit Check" is enabled, the current "Fault Limit" forms a new upper line segment replacing other previously defined limit lines (see Section 5.6.1.2, "Define Limit Lines dialog", on page 488).

Remote command:
`CALCulate<Chn>:TRANsform:DTFault:PEAK:STATe`

Fault Limit
Determines the level above which peaks in the active "Distance to Fault" trace are considered as faults.

The fault limit is defined relative to the 0 dB-line in the test diagram, i.e. the peak response value for total reflection after proper calibration.

Note:
- When the "Fault Limit" is modified, the new "Fault Limit" replaces other limit lines (see Section 5.6.1.2, "Define Limit Lines dialog", on page 488)
- This field is only enabled, if Fault Limit Check is active.

Remote command:
`CALCulate<Chn>:TRANsform:DTFault:PEAK:THReshold`

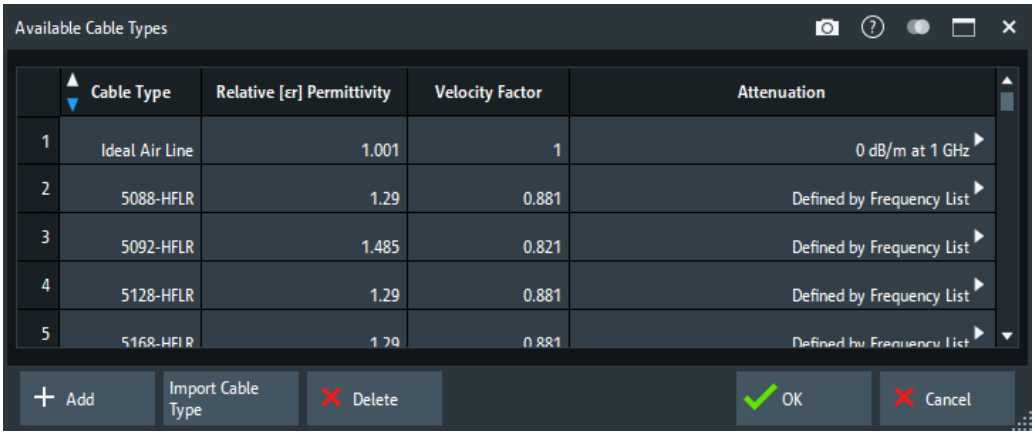
Fault List
This button is only enabled, if Fault Limit Check is active.

It opens the Fault List dock widget.

Available Cable Types dialog

The "Available Cable Types" dialog allows you to display predefined and user-defined cable types and to add/modify/delete user-defined cable types.

Among the predefined cable types, there is an ideal air line (with a relative permittivity of 1, a corresponding velocity factor of 1, and zero attenuation) and a wide range of standard cable types.



Cable types are defined by their relative permittivity *εr* or, equivalently, by their velocity factor *1/sqrt(εr)*, and a frequency-dependent attenuation table (see "Frequency-dependent attenuation table" on page 371).

The velocity factor is a measure for the velocity of an electromagnetic wave in a dielectric with permittivity ε_r, relative to its velocity in a vacuum (velocity factor < 1). It is needed to convert the propagation times into distances.

Frequency-dependent attenuation table

The "Attenuation of <Cable Type>" dialog defines the frequency dependence of the cable attenuation. The measured cables are assumed to be homogeneous so that the attenuation per length unit is constant. This means that the attenuation can be expressed in units of dB/m. Due to the skin effect and various other factors, the attenuation is frequency-dependent.

For predefined cables, the table is read-only.

Access: Available Cable Types dialog > "Attenuation" column > **...**



Given the specified attenuation values the R&S ZNB calculates the attenuation factor at the center of the channel's sweep range and corrects the impulse response trace using this attenuation factor *Attenuation (f_{center})*.

The frequency dependence can be defined in two alternative ways:

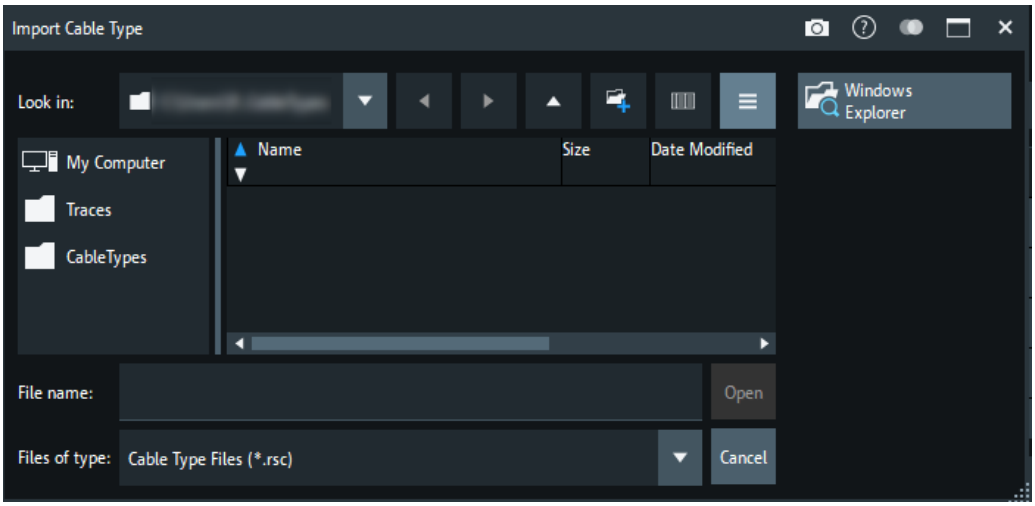
- **Single reference frequency**
  If the cable attenuation is specified at a single frequency f_1, the R&S ZNB calculates *Attenuation (f_{center})* using
  Attenuation (f_1) / Attenuation (f_{center}) = sqrt (f_1 / f_{center}),
  which accounts for the impact of the skin effect.
- **Frequency list**
  If the cable attenuation is specified at several frequency points, the R&S ZNB calculates the *Attenuation (f_{center})* by linear interpolation. If f_{center} is below (above) the specified frequency range, then *Attenuation (f_{center})* is linearly extrapolated from the two lowest (highest) frequency points.

Specify the cable attenuation as accurately as possible. The correction due to the attenuation factor is proportional to the measured distance between the fault and the reference plane: The larger the distance, the larger the correction.

Add / Delete
Creates/deletes a user-defined cable type.

Remote command:
`CALCulate<Chn>:TRANsform:DTFault:DEFine`
`CALCulate<Chn>:TRANsform:DTFault:DELete`
`MMEMory:LOAD:CABLe`
`MMEMory:STORe:CABLe`

Import Cable Type
Opens the "Import Cable Type" variant of the Open File dialog that lets you select a cable type file (`*.rsc`) for import.



Remote command:
`MMEMory:LOAD:CABLe`

Fault List dock widget

The "Fault List" dock widget allows you to explore and save all peaks that violate the active Fault Limit.

Access: [Trace] > "Distance to Fault" > "Fault List"

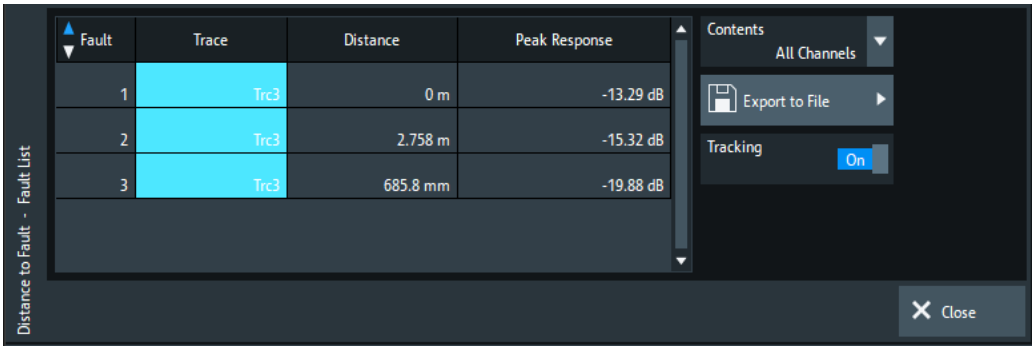


Figure 5-7: Fault List dock widget

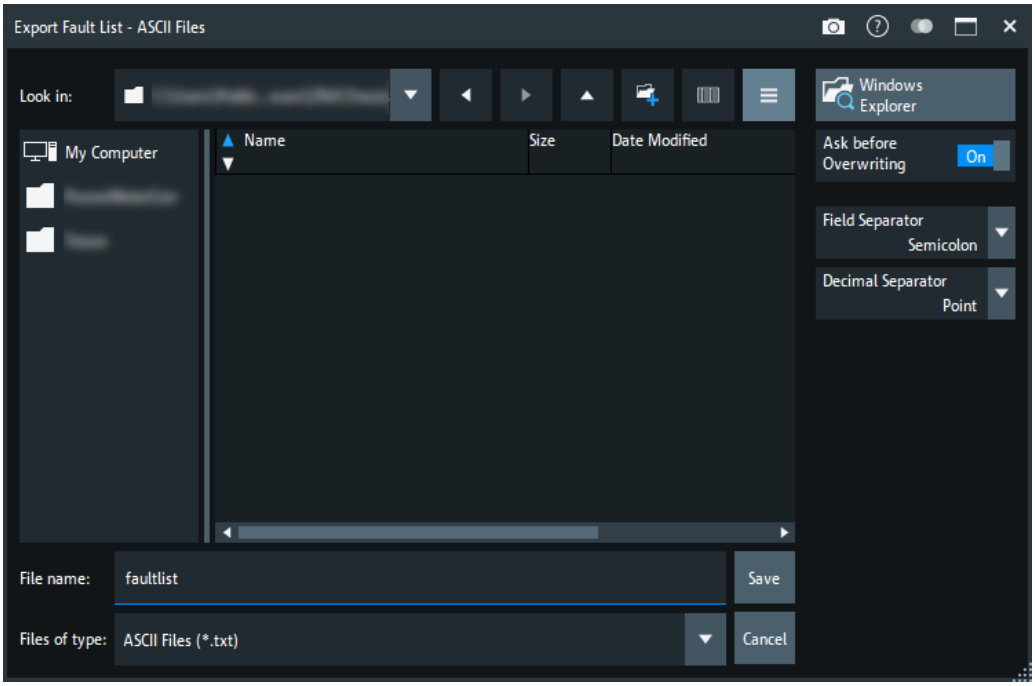
Fault Table
The displays a list of all peaks that violate the active Fault Limit. Unless tracking is disabled (see "Tracking Enabled" on page 374), the table is updated with every sweep.

Remote command:
`CALCulate<Chn>:TRANsform:DTFault:PEAK:COUNt?`
`CALCulate<Chn>:TRANsform:DTFault:PEAK:DATA<FaultNo>`

Contents
Allows you to filter the Fault Table:

- "All Channels" (default)
  The table displays the faults of all Distance to Fault traces in the current recall set.
- "Active Channel"
  The table displays the faults of all Distance to Fault traces in the active channel (if any).
- "Active Trace"
  The table displays the faults of the active trace (in case it is a Distance to Fault trace) .

Export to File
Opens a dialog that allows to export the (filtered) content of the fault list to an ASCII file with configurable "Field Separator" and "Decimal Separator".



This CSV type file contains one line (terminated by CR+LF) per fault. With ";" as "Field Separator" and "." as "Decimal Separator", the fault list of Figure 5-7 is exported to

```
1; Trc1;  457.2 mm; -8.56 dB
2; Trc1; 1.539 m; -9.24 dB
```

For an explanation of the possible filters, see "Contents" on page 373).

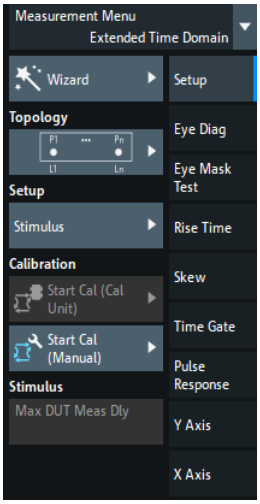
Tracking Enabled
If enabled (default), the "Fault Table" on page 373 is cleared at every sweep start and populated in real time. Disable "Tracking Enabled" to freeze the table contents.

5.2.6 Extended Time Domain

"Extended Time Domain" measurements are enabled by the extended time domain analysis option R&S ZNB-K20. They are only visible if this option is available.

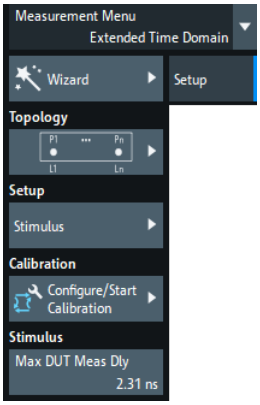
Access: Trace – [Meas] > "Measurement Menu" > "Extended Time Domain"

Background information
See Section 4.7.1.7, "Extended time domain analysis", on page 226.



5.2.6.1 TDR Setup tab

The "TDR Setup" tab allows you to set up the measurement for the time domain transformation. It is then used to analyze the time domain behavior of the DUT (eye diagram, rise time, skew, ...).



The "TDR Setup" softtool contains the following sections/controls:

- "TDR Wizard"
  Opens the Extended Time Domain Wizard.
- "Topology"
  Opens the "Balanced Ports" dialog that allows to configure the logical DUT ports (see "Balanced Ports dialog" on page 296)
- "Stimulus..."
  Opens the TDR Stimulus Settings dialog that allows to configure the frequency sweep whose results are then used for the time domain transformation.
- "Calibration"
  The "Calibration" section provides easy access to (basic) manual and automatic calibration functions (see Section 5.11.1, "Start Cal tab", on page 562)
- "Stimulus"

The "Stimulus" section provides quick access to settings that are also available in the TDR Stimulus Settings dialog. These settings are only available in Automatic Harmonic Grid mode, which is activated automatically when opening the "TDR Stimulus Settings" dialog.

5.2.6.2 Extended Time Domain Wizard

With the "Extended Time Domain Wizard", a non-RF engineer such as an electrical or signal integrity engineer can quickly set up and calibrate an extended time domain measurement. It comprises the following steps:

- Step 1: Port Topology
- Step 2: Stimulus Settings

Access: TDR Setup tab > "Wizard"

Step 1: Port Topology

The "Port Topology" step allows you to set up the (balanced) port configuration, similar to the Balanced Ports dialog.



Figure 5-8: Port Topology step (≤4 physical ports)

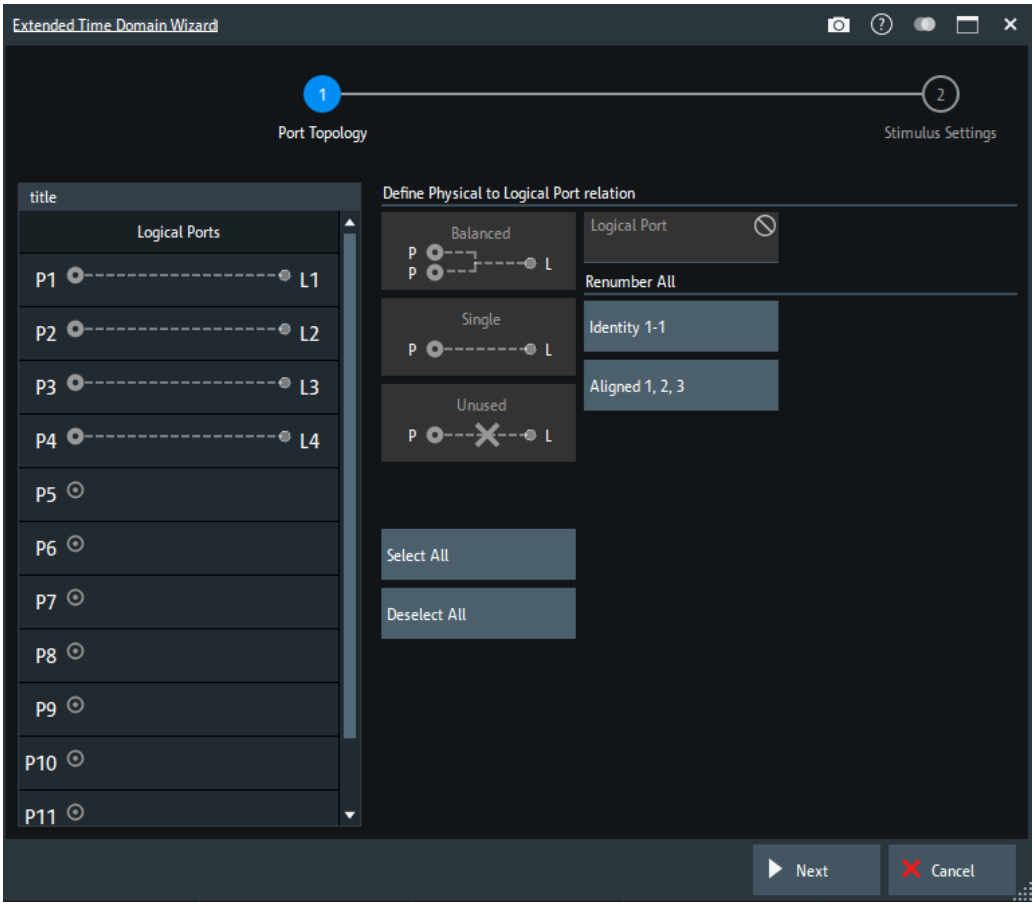


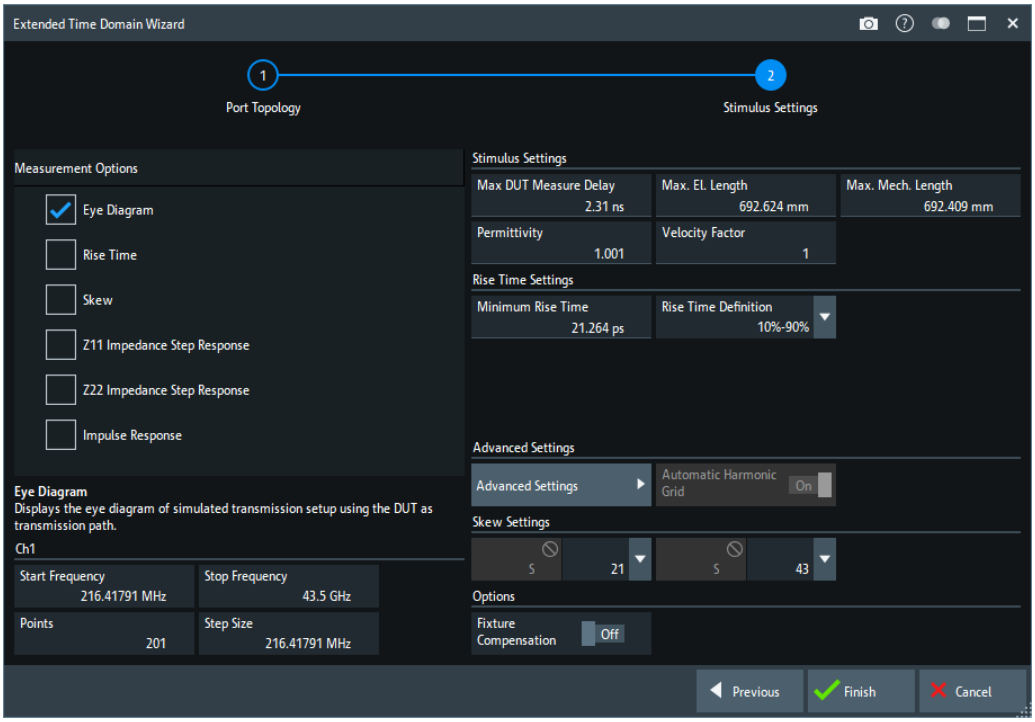
Figure 5-9: Port Topology step (> 4 physical ports)

Select "Next" to proceed to Step 2: Stimulus Settings.

Step 2: Stimulus Settings

The "Stimulus Settings" step gives access to the basic settings of the TDR Stimulus Settings dialog and allows you to select the time domain measurements to be performed.

The "TDR Wizard" converts the time domain test parameters to frequency domain settings.



Select a measurement to activate the "Finish" button. Activate "Fixture Compensation" if you want to proceed to the Fixture Compensation dialog.

5.2.6.3 TDR Stimulus Settings dialog

This dialog allows you to configure the frequency sweep whose results are then used for the time domain transformation and subsequent TDR simulation.

Access: TDR Setup tab > "Stimulus"

After the settings are made, the resulting sweep parameters are shown in the lower part of the dialog, as can be seen in the screenshot below.

Opening the dialog automatically activates the Automatic Harmonic Grid function.

Controls in the TDR Stimulus Settings dialog

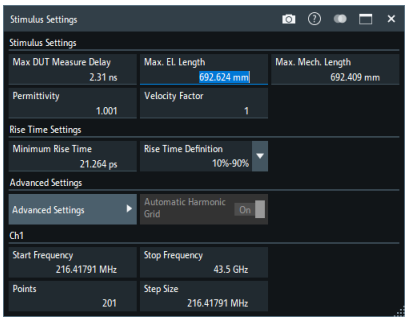


Figure 5-10: Stimulus Settings dialog

The "TDR Stimulus Settings" dialog offers basic settings. Instead of specifying the sweep parameters directly, they are derived from time domain properties of the DUT. After the settings are made, the resulting sweep parameters are shown in the dialog.

Advanced settings can be accessed via "Advanced Settings" (see "TDR Stimulus - Advanced Settings dialog" on page 380).

Max. DUT Measure Delay / Max. El. Length / Max. Mech. Length
Defines the "length" of the DUT, which can either be specified as delay, electrical length, or mechanical length.

Given the DUT's permittivity ε_r, these alternative "length" parameters are related by:

$$Delay = \frac{L_{mech} \cdot \sqrt{\varepsilon_r}}{c};\ Electrical\ Length = L_{mech} \cdot \sqrt{\varepsilon_r}$$

Hence, setting one of the "length" parameters determines the other two.

Note: The length of the DUT determines the appropriate start frequency and step size of the frequency sweep. It is important that the user enters a value that is at least as high as the actual one. A higher value is acceptable as long as the resulting start frequency is not below the analyzer's minimum frequency.

Remote command:
`[SENSe<Ch>:]HARMonic:DLENgth:DATA`
`[SENSe<Ch>:]HARMonic:ELENgth:DATA`
`[SENSe<Ch>:]HARMonic:MLENgth:DATA`

Permittivity / Velocity Factor
Specifies the (relative) permittivity ε_r of the DUT or, alternatively, its velocity factor 1/ sqrt(ε_r).

The velocity factor is a measure for the velocity of an electromagnetic wave in a dielectric with permittivity ε_r, relative to its velocity in a vacuum (velocity factor < 1).

Permittivity and velocity factor are coupled parameters, i.e. setting one of them determines the other. A higher permittivity implies a smaller mechanical length, but leaves delay and electrical length unchanged (see Max. DUT Measure Delay / Max. El. Length / Max. Mech. Length).

Remote command:
[SENSe<Ch>:]HARMonic:PERMittivity:DATA
[SENSe<Ch>:]HARMonic:VELocity:DATA

Rise Time Definition
Selects between rise time definitions 10%-90% (default) and 20%-80%.

Remote command:
[SENSe<Ch>:]HARMonic:RTIMe:THReshold

Rise Time
The minimum rise time (according to the selected Rise Time Definition) the user wishes to measure on the DUT.

The minimum rise time, in turn, determines the stop frequency of the frequency sweep: the smaller the "Rise Time", the larger the stop frequency – limited by the analyzer's maximum frequency.

Remote command:
[SENSe<Ch>:]HARMonic:RTIMe:DATA

TDR Stimulus - Advanced Settings dialog

The TDR Stimulus - Advanced Settings dialog compiles advanced settings of the time domain transformation that is available with the standard time domain option R&S ZNB-K2.

Access: TDR Stimulus Settings dialog > "Advanced Settings"

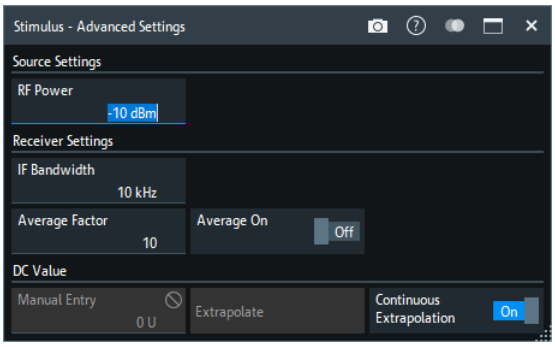


Figure 5-11: TDR Stimulus - Advanced Settings dialog

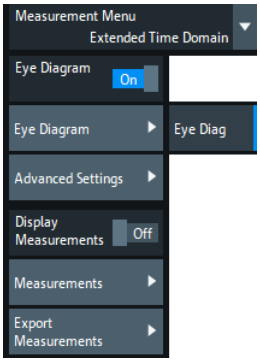
For additional settings, see Section 5.2.5.1, "Time Domain tab", on page 356.

Source/Receiver Settings
The "RF Power" is the output power of the R&S ZNB, "IF Bandwidth" and "Average Factor" determine the operation of the related R&S ZNB receiver. For a description and related remote control commands, see Section 5.9, "Power Bw Avg softtool", on page 531.

DC Value
Defines how to calculate the DC gain of the DUT (since the R&S ZNB cannot measure down to 0 Hz). These parameters are the same settings as for the standard time domain option R&S ZNB-K2. For a description and related remote control commands, see "DC Value" on page 362.

5.2.6.4 Eye Diag tab

The "Eye Diag" tab is for enabling and configuring the eye diagram and its measurements.



Eye Diagram
Enables the eye diagram with the current settings (default or defined in Eye Diagram dialog) or disables it.

Remote command:
`CALCulate<Chn>:EYE:STATe`

Eye Diagram ...
Opens the Eye Diagram dialog that allows to create an eye diagram and to perform basic settings on the simulated digital signal generator.

Advanced Settings...
If the active trace is an eye diagram, the "Advanced Settings..." button opens the Advanced Settings dialog that allows in-depth configuration of the simulated digital signal generator and receiver.

Display Measurements
If the active trace is an eye diagram, the "Display Measurements" button in the Eye Diag tab softtool tab toggles the display of the eye diagram results.

The available results depend on the selected Modulation:

- For NRZ modulated generator signals, the result display comprises two separate info fields for "Basic" and "Time" results (see "Measurements..." on page 382). By default they are stacked below each other, however, they can be moved independently like any other info field.



Figure 5-12: Eye Diagram Results (NRZ Modulation)

- For PAM modulation formats, only a reduced set of "Basic" results is available.



Figure 5-13: Eye Diagram Results (PAM Modulations)

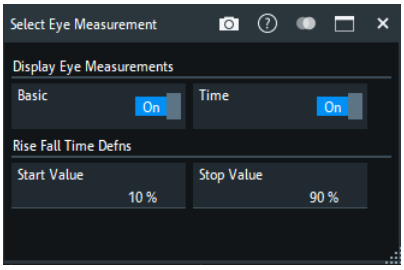
For a description of the result values, see "Eye diagram results" on page 227.

Remote command:

```
CALCulate<Chn>:EYE:MEASurement:STATe
CALCulate<Chn>:EYE:MEASurement:DATA?
```

Measurements...

If the active trace is an eye diagram, the "Measurements..." button brings up a dialog that allows to select and configure the eye measurements to be displayed (if Display Measurements is enabled).



The following measurement results can be selectively enabled/disabled:

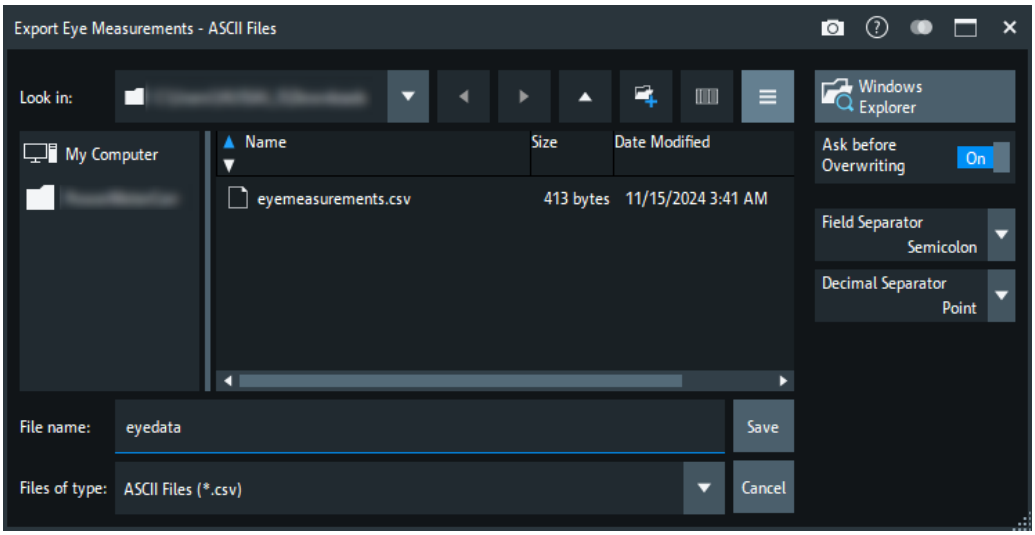
- "Basic" – Eye Minimum, Eye Maximum, Eye Base, Eye Top, Eye Mean, Eye Amplitude, Eye Height, Eye Width
  For PAM signals (see "Modulation" on page 385) Eye Height and Eye Width are not available.
- "Time" – Bit Period, Rise Time, Fall Time, Jitter Pk-Pk, Jitter RMS, Duty Cycle Dist, Duty Cycle Pct, Crossing Percent, Opening Factor, SNR
  For PAM signals these results are not available.

For a description of the result values, see "Eye diagram results" on page 227.

Furthermore it is possible to change the Rise (and Fall) Time definition for the eye measurement from the standard 10–90% (90–10%) step to any other "Start Value"/"Stop Value" pair.

Remote command:
CALCulate<Chn>:EYE:MEASurement:BASic
CALCulate<Chn>:EYE:MEASurement:TIME
CALCulate<Chn>:EYE:MEASurement:TTIMe:THReshold

Export Measurements...
If the active trace is an eye diagram, the "Export Measurements" button opens the "Export Eye Measurements – ASCII Files" dialog that allows to save the Eye diagram results to an ASCII file.



The "Export Eye Measurements – ASCII Files" dialog is a standard "Save File" dialog with the additional options of selecting a "Field Separator" (semicolon, comma, tab or space) and a "Decimal Separator" (comma or point).

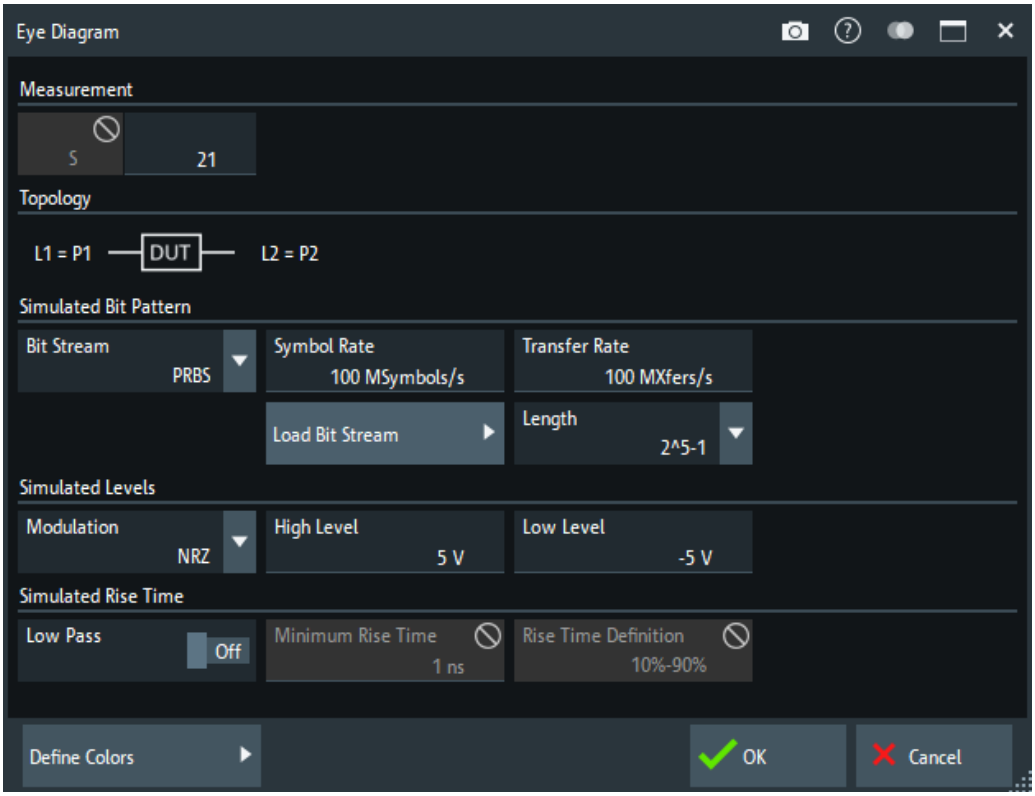
Note that the decimal separator and field separator must be different.

Remote command:
MMEMory:STORe:EYE:MEASurements

5.2.6.5 Eye Diagram dialog

Provides basic configuration of the eye diagram simulation.

Access: Eye Diag tab > "Eye Diagram"



For advanced configuration of the eye diagram simulation, see Section 5.2.6.6, "Advanced Settings dialog", on page 387.

Measurement / Topology
"Measurement" allows you to select an S-parameter, i.e. the transmission (or reflection) whose time domain properties you want to analyze. The "Topology" to the right is updated accordingly.

Remote command:
`CALCulate<Ch>:PARameter:MEASure`

Bit Stream
Describes the bit stream generated by the "virtual" generator.

"Bit Stream" can either be a pseudo random binary sequence (PRBS) of the selected Length or a user-defined bitstream.

The PRBSs are generated using a linear feedback shift register with generator polynomials allowing the maximal run length of the sequence.

Remote command:
`CALCulate<Chn>:EYE:INPut:BPATtern:TYPE`

Length
Length of the bit stream.

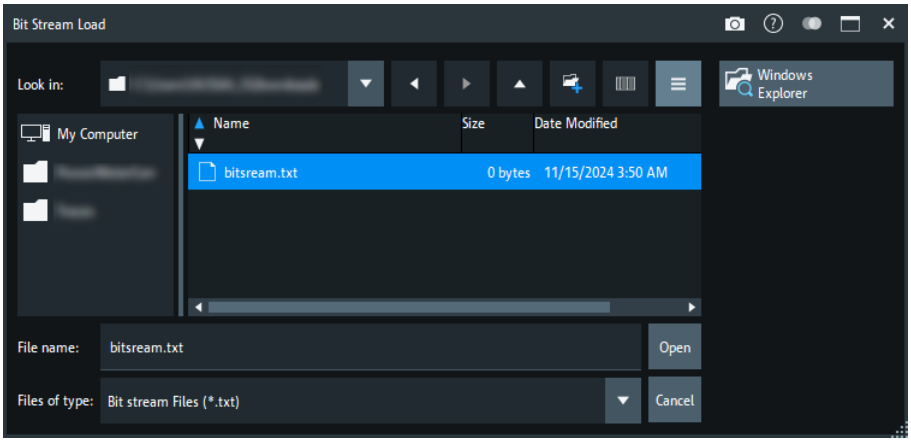
For a user-defined Bit Stream, the length can be specified in terms of bits, Dibits (2^10 bits), Mibits (2^20 bits), Gibits (2^30 bits).

Remote command:
CALCulate<Chn>:EYE:INPut:LENGth:PRBS
CALCulate<Chn>:EYE:INPut:LENGth:BITS

Load Bit Stream
A user-defined bitstream can be loaded from a file and is repeated until the configured Length is reached. If no pattern is loaded from a file, the default pattern "10" is repeated instead.

The "Bit Stream Load" dialog allows you to select a suitable "bit stream file".

Access: Eye Diagram dialog > "Load Bit Stream"



Such a bitstream file must be a 7-bit ASCII compatible text file; the Byte Order Mark (BOM) that is common with UTF encodings is not allowed. Only the ASCII codes 0x30 (“0”) and 0x31 (“1”) are interpreted, all other codes including whitespace and line endings are ignored.

Remote command:
MMEMory:LOAD:EYE:BPATtern

Symbol Rate/Transfer Rate
The **"Symbol Rate"** (= baud rate) defines the number of symbol changes per unit time, or, equivalently, the symbol period (= 1 / "Symbol Rate").

To convert symbol rate to data rate, multiply the symbol rate with the number of bits per symbol (1 bit per symbol for NRZ, ..., 4 bits per symbol for PAM-16).

The **"Transfer Rate"** is the number of transfers per second ("Xfers/s"). It can be calculated from the "Symbol Rate" and vice versa:

- For NRZ, transfers and symbols are identical.
- For PAM-*n*, the number of transfers is *n* x the number of symbols.

Remote command:
CALCulate<Chn>:EYE:INPut:DRATe
CALCulate<Chn>:EYE:INPut:XRATe

Modulation
Defines the modulation of the generated bitstream (NRZ, PAM-4, PAM-8 or PAM-16).

Note that for a modulation format other than NRZ, eye masks and eye mask tests are not available (the buttons on the Eye Mask Test tab are grayed out).

Remote command:
`CALCulate<Chn>:EYE:INPut:MODulation`

High Level / Low Level
Defines the highest/lowest (nominal) voltage level of the multilevel signal that is used to generate the related eye diagram.

The Modulation type can only be changed in the advanced "Generator Settings" dialog (see "Generator" on page 387).

Remote command:
`CALCulate<Chn>:EYE:INPut:OLEVel`
`CALCulate<Chn>:EYE:INPut:ZLEVel`

Low Pass
Defines the signal shape of the simulated digital signal: toggles between ideal rectangular shape ("Low Pass" = disabled, default) and a more realistic shape ("Low Pass" = enabled).

If enabled a single pole lowpass filter is inserted into the simulated signal path, which is defined using its rise time (see "Rise Time / Rise Time Definition" on page 386).

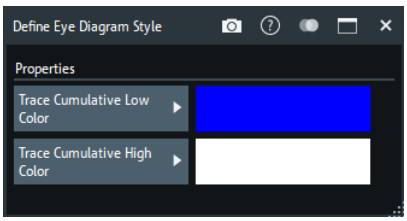
Remote command:
`CALCulate<Chn>:EYE:STIMulus:LOWPass`

Rise Time / Rise Time Definition
The Low Pass is defined by its "Rise Time" from 10% to 90% or from 20% to 80% of the signal amplitude.

Note that the measured rise time after the DUT is also affected by the rise time of the DUT. Hence the measured rise time can be higher than the specified rise time.

Remote command:
`CALCulate<Chn>:EYE:INPut:RTIMe:DATA`
`CALCulate<Chn>:EYE:INPut:RTIMe:THReshold`

Define Colors
Opens a dialog that allows to change the color gradient of the eye diagram (as a heat map).



- "Trace Cumulative High Color" is the color that is used for the most frequent occurrences.
- "Trace Cumulative Low Color" is the color that is used for the occurrence value 1.

No occurrence (value 0) is always displayed fully transparent with the background being visible.

Remote command:
DISPlay:CMAP:EYE:TCHigh:RGB
DISPlay:CMAP:EYE:TCLow:RGB

OK / Cancel
"OK" enables/modifies the eye diagram, "Cancel" leaves the Eye Diagram dialog without applying possible changes.

Remote command:
CALCulate<Chn>:EYE:STATe

5.2.6.6 Advanced Settings dialog

The "Advanced Settings" dialog gives full access to the calculation chain of the eye diagram simulation.

Access: Eye Diag tab > "Advanced Settings"

Controls in the dialog



- Generator, Emphasis, Jitter and Noise allow you to specify the simulated input signal of the DUT
- DUT allows you to switch between different transmission (and reflection) paths of the DUT and to configure its DC properties.
- Equalization allows you to perform signal equalization at the "virtual receiver".

A green LED indicates that the respective building block is active. However the calculation chain can also be shortened using the [Slider] control below the building blocks.

[Slider]
Allows you to shorten the simulation/calculation chain without deactivating the building blocks at the tail end.

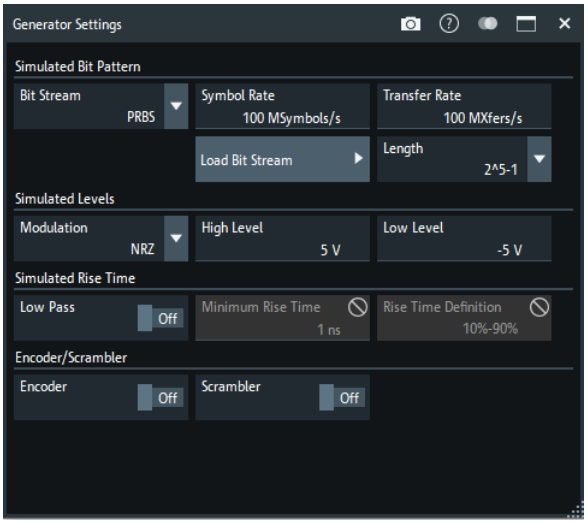
Remote command:
CALCulate<Chn>:EYE:VIEW

Define Colors...
See "Define Colors" on page 386.

Generator

The "Generator Settings" dialog gives full access to the "virtual" signal generator of the related eye diagram simulation. Most of the settings are also available in the Eye Diagram dialog.

Access: Advanced Settings dialog > "Generator"



Simulated Bit Pattern / Simulated Rise Time
See Section 5.2.6.5, "Eye Diagram dialog", on page 383.

Modulation
See "Modulation" on page 385.

High Level / Low Level
See "High Level / Low Level" on page 386.

Encoder
Enables or disables 8b/10b encoding of the original bitstream.

Remote command:
`CALCulate<Chn>:EYE:STIMulus:ENCoder`

Scrambler
Enables/disables scrambling of the (encoded) bitstream.

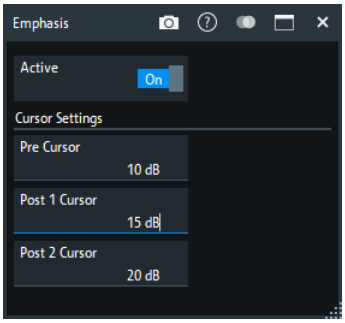
The scrambler is a linear feedback shift register (LFSR) implementing the polynomial G(X) = X^16+X^5+X^4+X^3+1.

Remote command:
`CALCulate<Chn>:EYE:STIMulus:SCRambler`

Emphasis

The "Emphasis" dialog allows you to introduce a pre-emphasis filter to the digital signal simulated for the eye diagram measurement. "Emphasis" and Equalization can be used to compensate for the signal integrity degradations caused by the DUT.

Access: Advanced Settings dialog > "Emphasis"

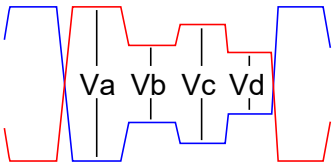


In transmission systems known losses of the channel are typically compensated already in the transmitter. The most common setting is to boost high frequencies compared to low frequencies since the channels show typically larger losses for high frequencies.

This type of equalization is done using finite impulse response (FIR) filters using between 2 and 4 taps. The filter's sampling time is identical to the bit period.

The coefficients of the FIR filter are not provided directly but in terms of pre- and post-cursor values. The cursor value is defined as the voltage ratio of two adjoining bits. Since up to four taps can be used, the bit pattern for defining all cursor values must use four consecutive bit values. Here a repeating bit pattern 0 1 1 1 1 0 is used. By definition the voltage levels for encoding the bits are symmetrical using +V for the ones and -V for the zeros.

In the graphic below the blue and the red trace indicate the signals used for differential encoding on a symmetric line. Va indicates the level directly after a transition, Vd indicates the level before a transition, Vb and Vc are optionally used to specify further levels after the transition.

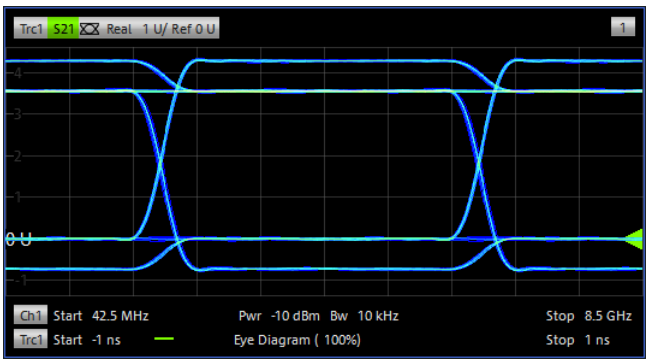


... 0 1 1 1 1 0 ...

In dB the pre- and post cursors are defined as:

- "Pre Cursor" = 20 $\log_{10}$(Vd / Vc)
- "Post 1 Cursor" = 20 $\log_{10}$(Vb / Va)
- "Post 2 Cursor" = 20 $\log_{10}$(Vc / Vb)

Simple applications of the emphasis only define the Post 1 Cursor. Use a negative value (in dB) for increasing the amplitude of the first bit after a transition. Below is an example for Post 1 Cursor = -3dB:



The FIR filter coefficients are found by solving the equations:

$c_{-1} + c_0 - c_1 - c_2 = Va$

$c_{-1} + c_0 + c_1 - c_2 = Vb$

$c_{-1} + c_0 + c_1 + c_2 = Vc$

$-c_{-1} + c_0 + c_1 + c_2 = Vd$

Subject to the normalization condition $| c_{-1} | + | c_0 | + | c_1 | + | c_2 | = 1$.

Active
Enables/disables pre-emphasis.

Remote command:
`CALCulate<Chn>:EYE:EMPHasis:STATe`

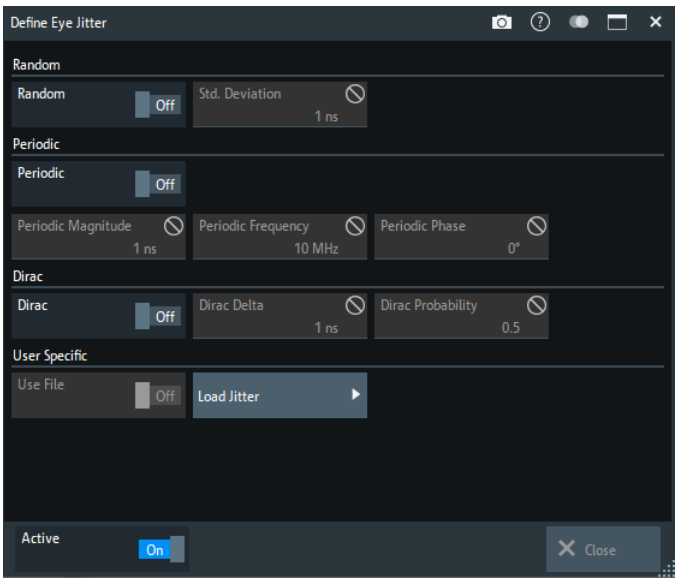
Cursor Settings
Sets the weights of the filter taps in the pre-emphasis FIR filter.

Remote command:
`CALCulate<Chn>:EYE:EMPHasis:CURSor:PRE`
`CALCulate<Chn>:EYE:EMPHasis:CURSor:POST<1|2>`

Jitter

The "Define Eye Jitter" dialog allows you to add jitter to the digital signal simulated for the eye diagram measurement.

Access: Advanced Settings dialog > "Jitter"



Four types of jitter can be configured and selectively enabled: random, periodic, Dirac and user-defined. If enabled, the jitter is added at the start of each symbol period, and its magnitude depends on the parameters specified in the dialog.

Active
Enable this button to activate any of the jitter sources. At this point, each of the individual jitter sources can be activated.

Remote command:
`CALCulate<Chn>:EYE:JITTer:STATe`

Random
Random jitter is determined from a normal (Gaussian) random distribution whose mean is zero, and whose "Std. Deviation" in seconds is specified by the user.

Remote command:
`CALCulate<Chn>:EYE:JITTer:TYPE:RANDom`
`CALCulate<Chn>:EYE:JITTer:RANDom:STDDeviation`

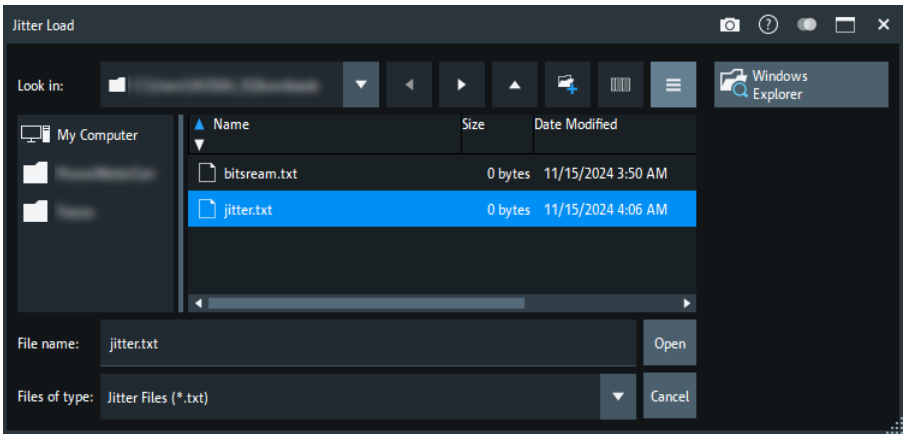
Periodic
This type of jitter is determined by a sine wave whose amplitude ("Periodic Magnitude"), frequency ("Periodic Frequency") and phase ("Periodic Phase") is specified by the user.

Remote command:
`CALCulate<Chn>:EYE:JITTer:TYPE:PERiodic`
`CALCulate<Chn>:EYE:JITTer:PERiodic:MAGNitude`
`CALCulate<Chn>:EYE:JITTer:PERiodic:FREQuency`
`CALCulate<Chn>:EYE:JITTer:PERiodic:PHASe`

Dirac
This type of jitter is specified by the amplitude in seconds ("Dirac Delta", positive or negative), and the probability of the jitter occurring at each symbol period ("Dirac Probability").

Remote command:
CALCulate<Chn>:EYE:JITTer:TYPE:DIRac
CALCulate<Chn>:EYE:JITTer:DIRac:DELTa
CALCulate<Chn>:EYE:JITTer:DIRac:PROBability

User Specific
The "Load Jitter" button allows the user to open a "Jitter File (*.txt)" with each entry in the file equal to the jitter to be added at each symbol period.



Such a Jitter File must be a 7-bit ASCII compatible text file; the Byte Order Mark (BOM) that is common with UTF encodings is not allowed. The file must consist of floating point values (in parsable format), separated by any whitespace and/or line endings.

Each value describes the jitter for a symbol transition (even when the current and the previous symbols are the same). The jitter value is implicitly given in the unit [s]. It denotes the delta between the time of the ideally expected transition (given by the data rate) and the actual one. Positive values describe a transition that is occurring after the ideally expected transition. The values are limited to the range -1.0e-3 to +1.0e-3. Any values outside this range are limited to this range.

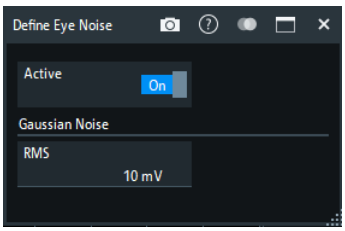
After the values in the file have been exhausted, the jitter values are taken starting from the beginning of the file.

Remote command:
CALCulate<Chn>:EYE:JITTer:TYPE:USER
MMEMory:LOAD:EYE:JITTer

Noise

The "Define Noise" dialog allows you to add Gaussian noise to the digital signal simulated for the eye diagram.

Access: Advanced Settings dialog > "Noise"



Active
Enables/disables noise insertion.

Remote command:
`CALCulate<Chn>:EYE:NOISe:STATe`

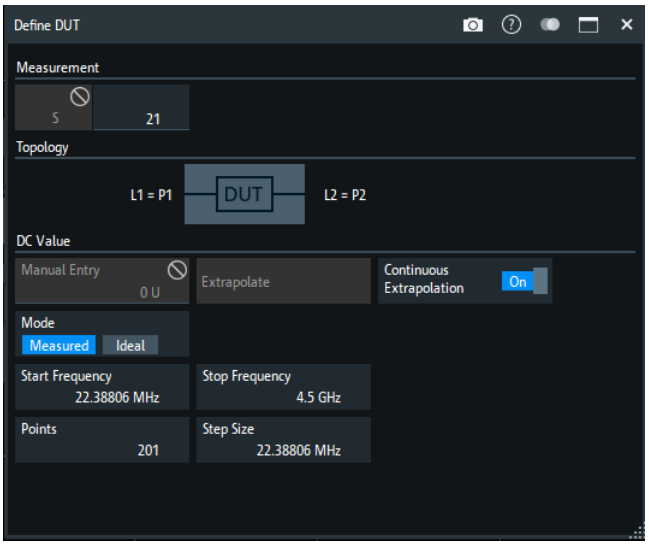
RMS
Defines the root mean square noise level.

Remote command:
`CALCulate<Chn>:EYE:NOISe:RMS`

DUT

The "Define DUT" dialog allows you to switch between different transmission (and reflection) paths of the DUT and to configure its DC properties. In addition, it allows you to switch between the real DUT (with measured frequency response) and an ideal one (with flat frequency response). This option can be useful when configuring the other eye diagram blocks (see Section 5.2.6.6, "Advanced Settings dialog", on page 387).

Access: Advanced Settings dialog > "DUT"



Measurement / Topolgy
See "Measurement / Topology" on page 384.

DC Value
See "DC Value" on page 362.

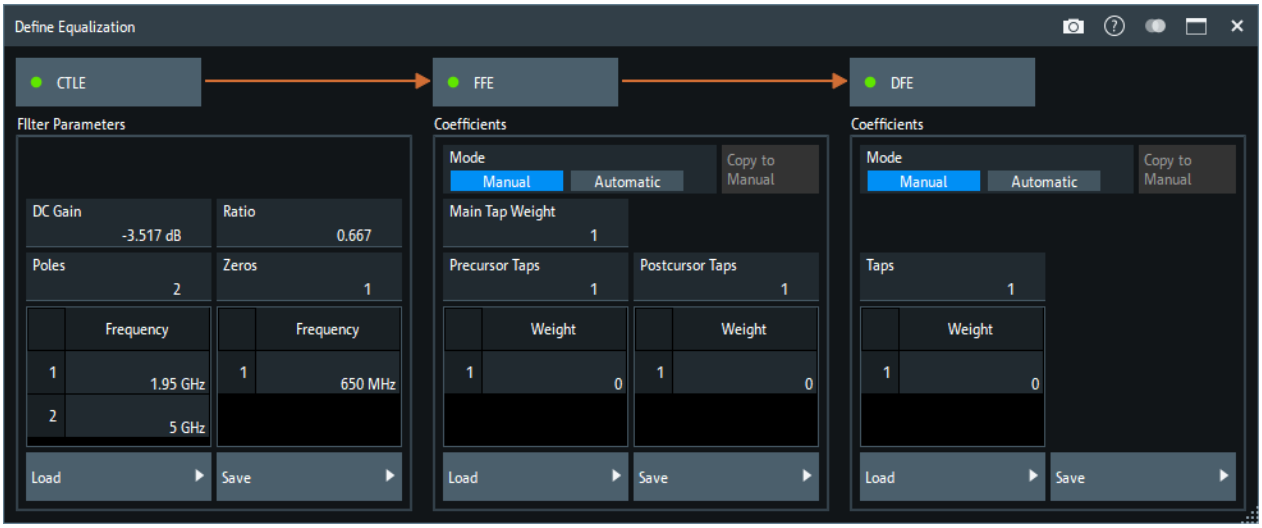
Mode
Allows you to switch temporarily between the real DUT (with measured frequency response) and an ideal one (with flat frequency response).

Remote command:
CALCulate<Chn>:EYE:DUT:MODE

Equalization

The "Define Equalization" dialog allows you to specify an equalizer at the receive side for the simulated eye diagram measurement. Similar to the usage of emphasis, the equalization targets to improve the signal quality at the receiver end of the transmission system.

Access: Advanced Settings dialog > "Equalize"



You can configure the following equalizers:

- CTLE: Continuous-time linear equalizer
- FFE: Feed forward equalizer
- DFE: Decision feedback equalizer

CTLE
The continuous-time linear equalizer is realized as a two-pole filter with a single zero. With angular frequencies ω = 2πf, the transfer function is given by:

$$H(s) = \frac{\text{"DC Gain"}\ \omega_{\text{Pole 1}}\ \omega_{\text{Pole 2}}}{\omega_{\text{Zero}}} \frac{s + \omega_{\text{Zero}}}{(s + \omega_{\text{Pole 1}})(s + \omega_{\text{Pole 2}})}$$

The default values are taken from the USB 3.0 standard for the 'Long Channel' model.

Remote command:
CALCulate<Chn>:EYE:EQUalization:CTLE:STATe

DC Gain, Ratio ← CTLE
Specifies the DC gain of the described transfer function (see above) in dB or as the ratio of output to input voltage at DC. Both parameters are coupled.

Remote command:
CALCulate<Chn>:EYE:EQUalization:CTLE:DC
CALCulate<Chn>:EYE:EQUalization:CTLE:RATio

Poles, Frequency ← CTLE
Indicates the number of poles of the filter function and lists the frequencies of the poles.

Remote command:
CALCulate<Chn>:EYE:EQUalization:CTLE:COUNt:POLE
CALCulate<Chn>:EYE:EQUalization:CTLE:LIST:POLE
CALCulate<Chn>:EYE:EQUalization:CTLE:POLE<Num>

Zeros, Frequency ← CTLE
Indicates the number of zeros of the filter function and lists the frequencies of the zeros.

Remote command:
CALCulate<Chn>:EYE:EQUalization:CTLE:COUNt:ZERO
CALCulate<Chn>:EYE:EQUalization:CTLE:LIST:ZERO
CALCulate<Chn>:EYE:EQUalization:CTLE:ZERO<Num>

Load, Save ← CTLE
Loads an equalizer specification from a `.json` file or saves the equalizer specification to a `.json` file.

Remote command:
MMEMory:LOAD:EYE:EQUalization:CTLE on page 1269

FFE
The feed-forward equalizer creates a pre-distorted pulse by combining the main pulse with weighted delayed pulses. It is also possible to include weighted advanced pulses. With proper weights of the additional pulses, the overall intersymbol interference (ISI) can be reduced. The main pulse is related to the main tap of the equalizer. The delayed pulses are indicated as postcursor taps, the advanced pulses as precursor taps.

Remote command:
CALCulate<Chn>:EYE:EQUalization:FFE:STATe

Manual, Automatic ← FFE
With "Manual" selected, you specify the number of taps and their weights (the tap coefficients). With "Automatic", the tap coefficients are computed automatically using a least squares zero-forcing algorithm. Switching between "Manual" and "Automatic" retains the manual weights set by the user.

Remote command:
CALCulate<Chn>:EYE:EQUalization:FFE:MODE
MMEMory:STORe:EYE:EQUalization:CTLE

Copy to Manual ← FFE
Only active in "Automatic" mode. Click this button to overwrite the weights set in "Manual" mode by the displayed "Automatic" weights. The mode is switched to "Manual".

Main Tap Weight ← FFE
The weight of the main pulse at the data sampling time.

Remote command:
CALCulate<Chn>:EYE:EQUalization:FFE:MAIN:TAP

Precurser Taps, Weight ← FFE
Specifies the number of precursor taps and their weights.

Remote command:
CALCulate<Chn>:EYE:EQUalization:FFE:COUNt:PRECursor
CALCulate<Chn>:EYE:EQUalization:FFE:LIST:PRECursor
CALCulate<Chn>:EYE:EQUalization:FFE:PRECursor<Num>

Postcurser Taps, Weight ← FFE
Specifies the number of postcursor taps and their weights.

Remote command:
CALCulate<Chn>:EYE:EQUalization:FFE:COUNt:POSTcursor
CALCulate<Chn>:EYE:EQUalization:FFE:LIST:POSTcursor
CALCulate<Chn>:EYE:EQUalization:FFE:POSTcursor<Num>

Load, Save ← FFE
Loads an equalizer specification from a `.json` file or saves the equalizer specification to a `.json` file.

Remote command:
MMEMory:LOAD:EYE:EQUalization:FFE
MMEMory:STORe:EYE:EQUalization:FFE

DFE
The decision feedback equalizer uses past symbol decisions and feeds them back to subtract previously detected interference from the current signal.

Remote command:
CALCulate<Chn>:EYE:EQUalization:DFE:STATe

Manual, Automatic ← DFE
With "Manual" selected, you specify the number of taps and their weights (the tap coefficients). With "Automatic", the tap coefficients are computed automatically. Switching between "Manual" and "Automatic" retains the manual weights set by the user.

Remote command:
CALCulate<Chn>:EYE:EQUalization:DFE:MODE

Copy to Manual ← DFE
Only active in "Automatic" mode. Click this button to overwrite the weights set in "Manual" mode by the displayed "Automatic" weights. The mode is switched to "Manual".

Taps, Weight ← DFE
Specifies the number of precursor taps and their weights.

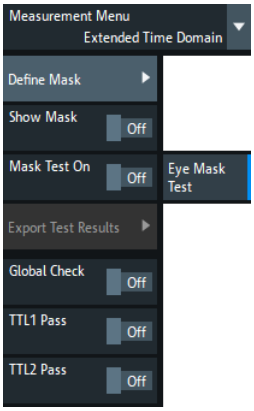
Remote command:
CALCulate<Chn>:EYE:EQUalization:DFE:COUNt:TAPS
CALCulate<Chn>:EYE:EQUalization:DFE:LIST:TAPS
CALCulate<Chn>:EYE:EQUalization:DFE:TAP<Num>

Load, Save ← DFE
Loads an equalizer specification from a `.json` file or saves the equalizer specification to a `.json` file.

Remote command:
MMEMory:LOAD:EYE:EQUalization:DFE
MMEMory:STORe:EYE:EQUalization:DFE

5.2.6.7 Eye Mask Test tab

If the active trace is represented as an eye diagram, the "Eye Mask Test" tab allows you to set up an eye mask for it. In this way, you enable testing against this mask and exporting the test results.



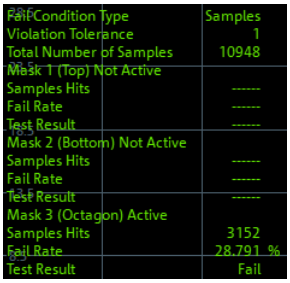
For a Modulation type other than NRZ, display of eye mask and eye mask test is not available, i.e. the buttons in the "Eye Mask Test" tab are grayed out.

Define Mask
Opens the Define Mask Configuration dialog.

Show Mask
Determines whether the mask is shown in the eye diagram display.

Remote command:
CALCulate<Chn>:EYE:MASK:SHOW

Mask Test On
Tells the firmware to run the mask test after every recalculation of the eye diagram. If the mask test is enabled, an info field with the test results is displayed:



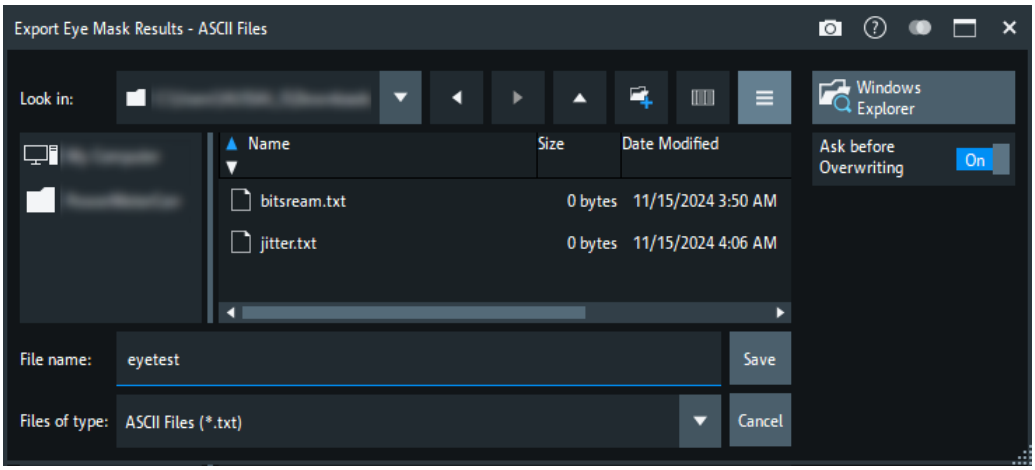
A TTL signal indicating pass or fail can be generated in addition.

Remote command:

```
CALCulate<Chn>:EYE:MASK:STATe
CALCulate<Chn>:EYE:MASK:DATA?
CALCulate<Chn>:EYE:MASK:FAIL?
```

Export Test Results

Opens a dialog that allows the user to save the mask test results to an ASCII file.



Remote command:

```
MMEMory:STORe:EYE:MASK:RESults
```

Global Check, TTL1 Pass, TTL2 Pass

See Section 5.6.1, "Limit Test tab", on page 484.

5.2.6.8 Define Mask Configuration dialog

Allows you to set up the mask the simulated eye diagram is tested against.

Access: System – [Applic] > "TDR" > "Eye Mask Test" > "Define Mask"

There are three mask areas that can be set up for the eye diagram test: center polygon, top rectangle, and bottom rectangle. Testing against these mask areas can be selectively enabled.



Polygon Setup
Allows you to define the center polygon and to activate it in the mask test. The center polygon can either be an octagon, a hexagon, or a rectangle ("Shape" combo-box). The sizes of the polygon can be set using the main/minor widths and heights as shown in the corresponding visualization.

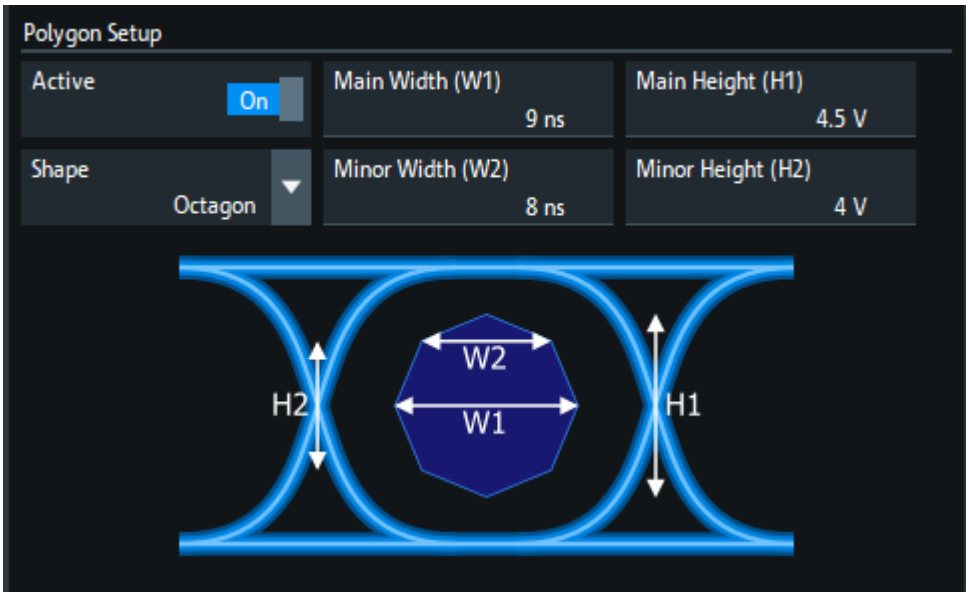


Figure 5-14: Center polygon setup: octagon



Figure 5-15: Center polygon setup: hexagon

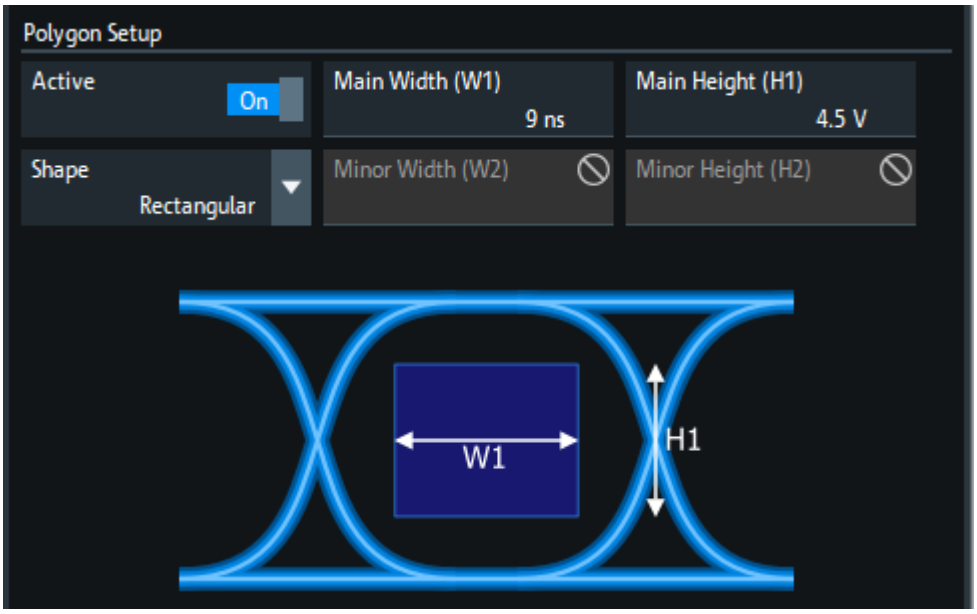


Figure 5-16: Center polygon setup: rectangle

The polygon is centered at the Mask Center.

Remote command:

```
CALCulate<Chn>:EYE:MASK:SHAPe:POLYgon:STATe
CALCulate<Chn>:EYE:MASK:SHAPe:POLYgon:TYPE
CALCulate<Chn>:EYE:MASK:SHAPe:POLYgon:HORizontal
CALCulate<Chn>:EYE:MASK:SHAPe:POLYgon:VERTical
```

Top/Bottom Setup
Defines and activates the top and bottom rectangles.

Both rectangles are horizontally centered at the Mask Center and also their vertical offsets are specified relative to this point.

Remote command:

```
CALCulate<Chn>:EYE:MASK:SHAPe:TOP:STATe
CALCulate<Chn>:EYE:MASK:SHAPe:TOP:HORizontal
CALCulate<Chn>:EYE:MASK:SHAPe:TOP:VERTical
CALCulate<Chn>:EYE:MASK:SHAPe:BOTTom:STATe
CALCulate<Chn>:EYE:MASK:SHAPe:BOTTom:HORizontal
CALCulate<Chn>:EYE:MASK:SHAPe:BOTTom:VERTical
```

Mask Center

Defines the center point of the eye mask:

- The center polygon is centered at this point.
- The top and bottom rectangles are horizontally centered at the mask center and also their vertical offsets are specified relative to this point.

Remote command:

```
CALCulate<Chn>:EYE:MASK:CENTer:HORizontal
CALCulate<Chn>:EYE:MASK:CENTer:VERTical
```

Test Settings

Allows you to set the absolute or relative number of mask violations that cause the mask test to fail.

Remote command:

```
CALCulate<Chn>:EYE:MASK:FAIL:CONDition
CALCulate<Chn>:EYE:MASK:VIOLation:TOLerance
CALCulate<Chn>:EYE:MASK:VIOLation:RATE
```

Save / Load Mask Configuration

Opens a dialog that allows to save/load the mask test configuration to/from a 7bit ASCII file (`*.mask`). An example is shown below:

```
[Mask Top]
Active = true
Width Top = 9e-009
Offset Top = 6

[Mask Bottom]
Active = true
Width Bottom = 9e-009
Offset Bottom = 6

[Mask Polygon]
Active = true
Shape = Octagon
Main Width = 9e-009
Minor Width = 8e-009
Main Height = 9
Minor Height = 8
```

```
[MASK CENTER]
Horizontal = 0
Vertical = 0

[TEST SETTING]
Fail Condition = Samples (Percent)
Violation Tolerance = 1
Violation Percent   = 10
```

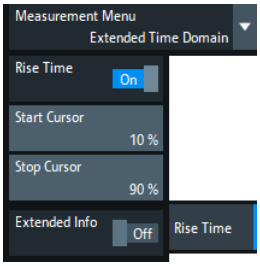
Remote command:
MMEMory:LOAD:EYE:MASK
MMEMory:STORe:EYE:MASK

Automatic Mask Generation
The "Automatic Mask Generation" function attempts to size the polygon, top, and bottom masks in a reasonable manner based on the current eye measurement settings.

Remote command:
CALCulate<Chn>:EYE:MASK AUTO

5.2.6.9 Rise Time tab

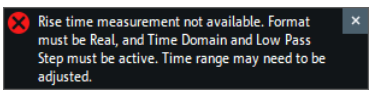
The "Rise Time" tab allows you to enable and configure the Rise time measurement.



Rise Time
Enables/disables the rise time measurement.

The rise time measurement can only be performed if the active trace is Real, Time Domain is enabled, and the transformation Type "Low Pass Step" is used. The latter, in turn, requires the stimulus grid to be harmonic.

If the rise time measurement cannot be enabled, a message is displayed:



Remote command:
CALCulate<Chn>:TTIMe:STATe

Start Cursor / Stop Cursor
Defines the lower/upper percentage for the rise time measurement.

Remote command:
CALCulate<Chn>:TTIMe:THReshold

Extended Info
Defines whether the info field for the rise time measurement only provides basic information (default) or extended information:

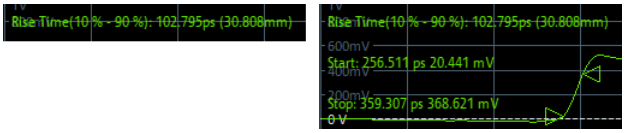
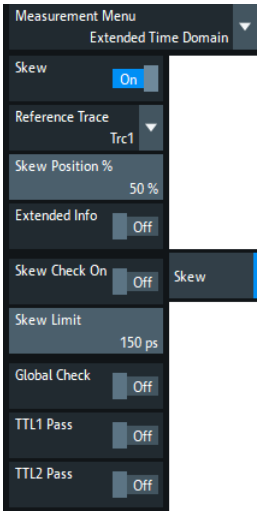


Figure 5-17: Rise time info, basic vs. extended

Remote command:
`CALCulate<Chn>:TTIMe:DATA`

5.2.6.10 Skew tab

The "Skew" tab allows you to enable and configure the Skew measurement.

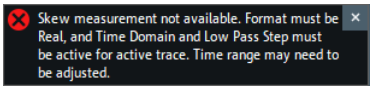


Skew
Enables/disables the skew measurement between the active trace and the Reference Trace

The skew measurement can only be performed if the following conditions are met for both the active trace and the reference trace:

- The trace format is Real.
- Time Domain is enabled.
- The "Low Pass Step" time domain transformation Type is used (which requires the stimulus grid to be harmonic)

If these conditions are not met, a message is displayed:



Remote command:
`CALCulate<Chn>:DTIMe:STATe`

Reference Trace
Selects the reference trace for the skew measurement.

Note that the reference trace must be in the same channel as the active trace.

See Skew for additional conditions on both the active and the reference trace.

Remote command:
`CALCulate<Chn>:DTIMe:TARGet`

Skew Position
Defines the percentage of the step size that defines the position of the reference points on the current and the reference trace. The "Skew" value is the delta between the X coordinates of these reference points.

Remote command:
`CALCulate<Chn>:DTIMe:POSition`

Extended Info
Defines whether the info field for the skew measurement only provides basic information (default) or extended information:

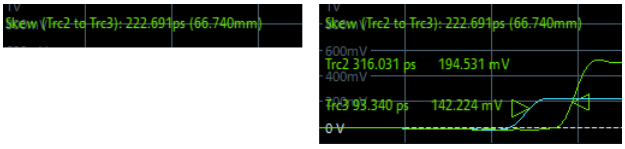


Figure 5-18: Skew info, basic vs. extended

The displayed skew values are positive, if, at the defined Skew Position, the active trace is to the right of the Reference Trace.

Remote command:
`CALCulate<Chn>:DTIMe:DATA?`

Skew Check On
Enables the skew check with the specified Skew Limit.

Remote command:
`CALCulate<Chn>:DTIMe:LIMit:STATe`

Skew Limit
Defines the limit value for the skew check. The unit depends on the current format of the x-axis (see "Time / Distance" on page 530).

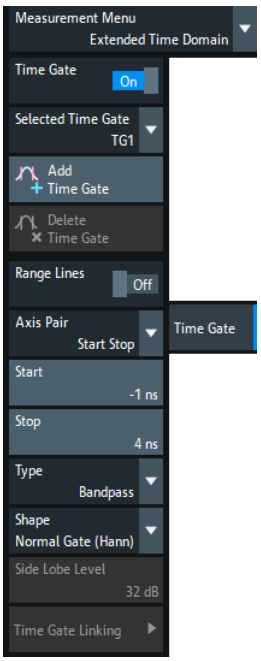
A TTL signal indicating pass or fail can be generated in addition.

Remote command:
`CALCulate<Chn>:DTIMe:LIMit:LIMit`

Global Check, TTL1 Pass, TTL2 Pass
See Section 5.6.1, "Limit Test tab", on page 484.

5.2.6.11 Time Gate tab

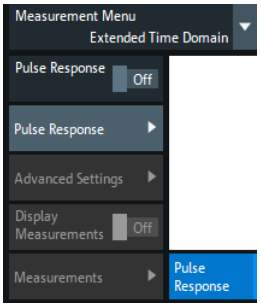
A copy of Section 5.2.5.2, "Time Gate tab", on page 363.



Since for the eye diagram, the time axis always equals two times the symbol time, the display of the time gate range lines ("Show Range Lines") does not make sense, and is therefore not available.

5.2.6.12 Pulse Response tab

Simulates pulses, which are transmitted to the DUT, measures an S-parameter and calculates the pulse response from the measurement. Key characteristics such as pre-cursors, main cursor, post-cursors, and intersymbol interference (ISI) become visible. Pulse response and ISI are critical in designing equalization filters to compensate signal loss.



Pulse Response
Enables the pulse response measurements based on the current settings.

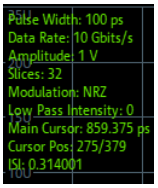
When pulse response is enabled, the following settings in time domain analysis are automatically applied: harmonic frequency grid, lowpass impulse response and real trace format. Therefore, it is recommended to use the preset function to restore the default settings for other measurements (e.g., eye diagram, rise time) when switching from pulse response.

Remote command:
`CALCulate<Chn>:PULSe:STATe`

Pulse Response ...
Opens the Pulse response dialog that specifies the measured quantity and the pulse simulation.

Advanced Settings ...
Opens the Pulse Response Advanced Settings dialog containing a sequence of "Generator", "Emphasis", "DUT" and "Equalizer" settings.

Display Measurements
If enabled, the main settings and measurement results including information on the main cursor and intersymbol interference (ISI) are displayed in the diagram.



Remote command:
`CALCulate<Chn>:PULSe:MEASurement:STATe`

Measurements
Opens the Pulse response Measurements dialog that configures the display measurements.

5.2.6.13 Pulse response dialog

Specifies the measured quantity and the pulse simulation.

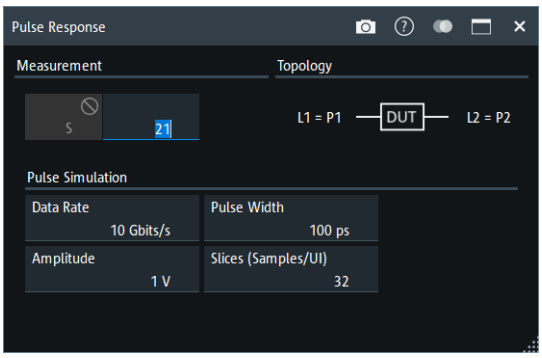


Figure 5-19: Pulse Response dialog

Access: Pulse Response tab > "Pulse Response"

Measurement, Topology
Specifies the quantity measured as pulse response, for example S21. The "Topology" to the right showing the mapping of logical ports to physical ports is updated accordingly.

Data Rate
The number of pulses generated per second.

Remote command:
`CALCulate<Chn>:PULSe:INPut:DRATe`

Pulse Width
The width of the generated pulses.

Remote command:
`CALCulate<Chn>:PULSe:INPut:PWIDth`

Amplitude
The maximum voltage value of the generated pulses.

Remote command:
`CALCulate<Chn>:PULSe:INPut:AMPLitude`

Slice (Samples/UI)
The number of samples per unit interval representing the data oversampling rate.

Remote command:
`CALCulate<Chn>:PULSe:INPut:SLICes`

5.2.6.14 Pulse Response Advanced Settings dialog

The dialog configures the simulated pulses, filters and equalizers and more in a sequence of steps. It is very similar to the "Advanced Settings" dialog in Eye Diagram dialog.



Access: Pulse Response tab > "Advanced Settings"

Generator
The "Generator" dialog has the same settings as the Pulse response dialog.

Emphasis
Configures the pre-emphasis filter to compensate for signal loss in the transmitter end. Defines also pre-cursor and post-cursor values used to generate the pre-emphasis filter.

For explanations, see "Emphasis" on page 388.

Remote command:
CALCulate<Chn>:PULSe:EMPHasis:PRE
CALCulate<Chn>:PULSe:EMPHasis:PST1
CALCulate<Chn>:PULSe:EMPHasis:PST2
CALCulate<Chn>:PULSe:EMPHasis:STATe

DUT
For explanations, see DUT.

Remote command:
CALCulate<Chn>:PULSe:DUT:MODE on page 1075

Equalization
Specifies the CTLE equalizer and activates or deactivates it.



For explanaions,see "CTLE" on page 394.

Remote command:
CALCulate<Chn>:EYE:EQUalization:STATe
CALCulate<Chn>:PULSe:EQUalization:CTLE:DC
CALCulate<Chn>:PULSe:EQUalization:CTLE:POL1
CALCulate<Chn>:PULSe:EQUalization:CTLE:POL2
CALCulate<Chn>:PULSe:EQUalization:CTLE:RATio
CALCulate<Chn>:PULSe:EQUalization:CTLE:ZERO
CALCulate<Chn>:PULSe:EQUalization:STATe

5.2.6.15 Pulse response Measurements dialog

This dialog specifies some details how the results of the pulse response measurement are displayed in the diagram.

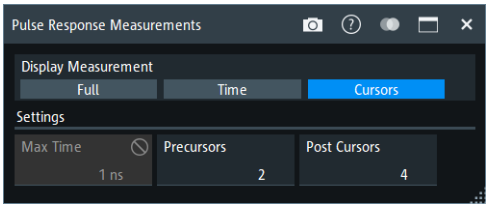


Figure 5-20: Pulse Response Measurements dialog

Full
If selected, all signals are included in the measurement results, resulting in the maximum value of intersymbol interference (ISI).

Remote command:
CALCulate<Chn>:PULSe:MEASurement:TYPE

Time, Max Time
If you select "Time" in the "Display Measurement" section, the pulse response is displayed over the "Max Time" specified in the "Settings" section. Hence, "Max Time" determines the range of included signals.

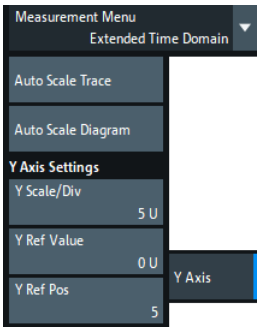
Remote command:
CALCulate<Chn>:PULSe:MEASurement:MAXTime on page 1080

Cursors, Precursors, Post Cursors
The cursors represent time samples of the pulse response spaced one unit interval (symbol width) apart. If you select "Cursors" in the "Display Measurement" section, you can select the numbers of "Precursors" and "Post Cursors" in the "Settings" section. These "Precursors" and "Post Cursors" are shown in the diagram, and the time range is adjusted for best representation of the cursors.

Remote command:
CALCulate<Chn>:PULSe:MEASurement:CURSors:PRE
CALCulate<Chn>:PULSe:MEASurement:CURSors:POST

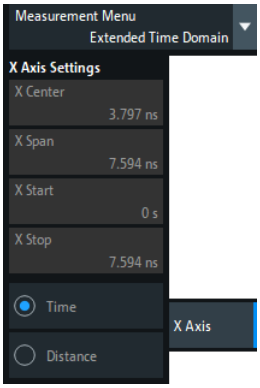
5.2.6.16 Y Axis tab

The "Y Axis" tab allows you to define the y-axis scaling of the active diagram. The scaling logic is the same as for all other diagram types, see Section 5.4, "Scale softtool", on page 444.



5.2.6.17 X Axis tab

The "X Axis" tab allows you to define the x-axis scaling of the active diagram. The scaling logic is the same as for all other diagram types, see Section 5.4, "Scale softtool", on page 444.



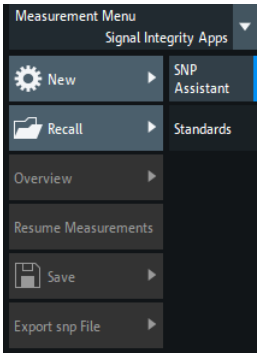
If the active trace is represented as an eye diagram, the "X Axis Settings" are grayed out, as shown above.

If the active trace is represented as a rise time or skew diagram: The "X Axis Settings" allow you to adjust the center/span or start/stop, no matter if the x-axis represents "Time" or "Distance" (see "Time / Distance" on page 530).

5.2.7 Signal Integrity Apps

Signal integrity measurements (user-defined or standardized)

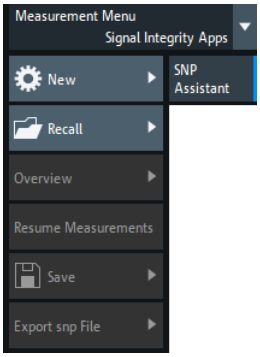
Access: Trace – [Meas] > "Measurement Menu: Signal Integrity Apps"



5.2.7.1 SNP Assistant tab

The SNP assistant (SNPA) is made available with software option R&S ZNB-K100. Its DUT centric approach to S-parameter-based characterization is particularly useful if the number of physical ports on the VNA and connected switch matrices is smaller than the number of physical DUT ports.

If option R&S ZNB-K100 is missing, the "SNP Assistant" application is not available.



Controls on the SNP Assistant tab

New...
Creates an SNPA project and opens it in the SNP Assistant DUT Topology view.

Remote command:
`[SENSe:]SNPMeasure:RST`

Recall...
Opens a file browser that allows you to load an existing SNPA project from file (`*.bvm`).

The SNPA immediately proceeds to the "SNP Assistant DUT Topology" view.

Note: Legacy SNPA project files (`*.snp`) can still be read.

Remote command:
`[SENSe:]SNPMeasure:RECall`

Overview...
Opens the SNP Assistant Overview. Disabled, if no SNPA project is loaded.

Resume Measurements
Opens the SNP Assistant Measurements dock widget that guides you through the measurements defined in the loaded SNPA project. Disabled, if no SNPA project is loaded.

Save...
Saves the current SNPA project to a `*.bvm` file. Disabled, if no SNPA project is loaded.

This file format is portable between all Rohde & Schwarz network analyzers supporting the SNPA feature (option K100). However, on a R&S ZNA it requires firmware version 3.0, and on a R&S ZNB3000, R&S ZNB, R&S ZNBT or R&S ZND it requires firmware version 4.0.

Note: An SNPA project not only comprises the DUT setup (topology, port numbering and labeling) and the measurement setup (scope, sweep settings, data state), but also the available data – no matter if measured or loaded from Touchstone file.

Remote command:
`[SENSe:]SNPMeasure:SAVE`

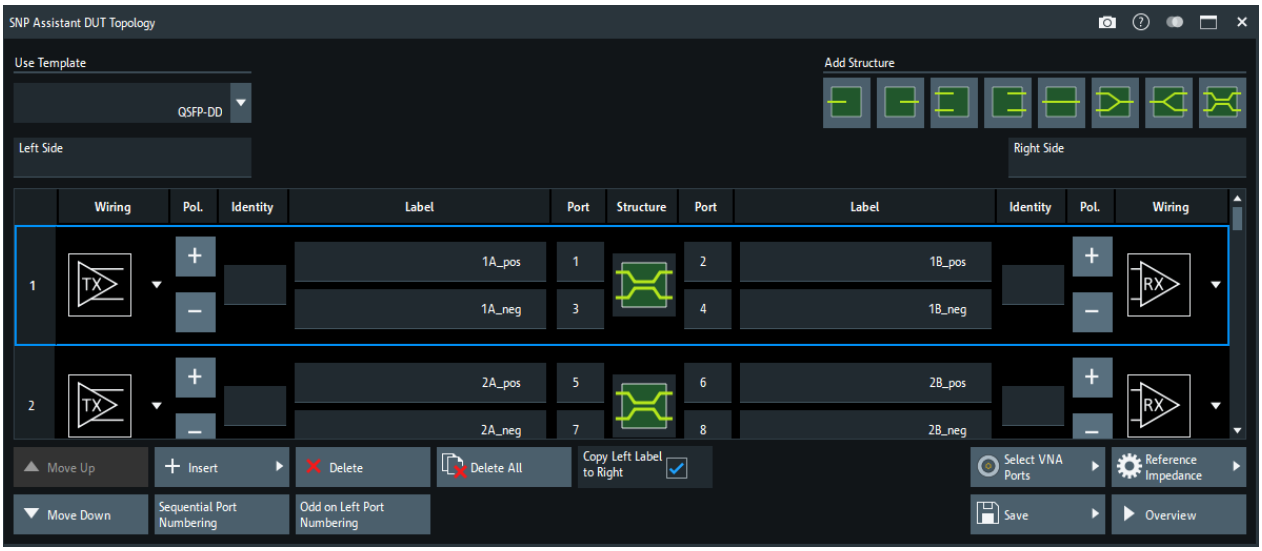
Export snp File...
Opens a file browser that allows you to export the S-parameter data (loaded or measured) of the SNPA project in Touchstone file format. This button is only enabled if none of the modeled DUT's S-parameters is in "Unmeasured" state. I.e. all S-parameters must be either "Measured", "Imported", or "Idealized".

Remote command:
`[SENSe:]SNPMeasure:EXPort`

SNP Assistant DUT Topology view

The "DUT Topology" view allows you to model your DUT, i.e.:

- Declare the (logical) DUT ports on the left and right, and the (intended) signal paths between them.
  - Ports can be balanced or single-ended.
  - Paths can be specified between any left-side and right-side port.
- Declare intended signal flow directions (left-to-right, right-to-left, bidirectional)



Add Structure
The buttons in the "Add Structure" section represent all possible combinations of port types (single-ended, balanced) and their connectivity (through-connected, terminated).

Table 5-3: DUT structures

Buttons	Description	SCPI enum constants
	Single-ended terminated structures	D1, D1R
	Balanced terminated structures (isolated physical ports)	D2D, D2DR

Buttons	Description	SCPI enum constants
	Single-ended and balanced through structures	D2 and D4
	Mixed mode through structures	D3, D3R

Clicking a button adds the corresponding structure to the DUT topology table.

Note: A maximum of 32 structures can be used to model a DUT.

Remote command:

```
[SENSe:]SNPMeasure:DUT:PART:ADD D1 | D1R | D2 | D2D | D2DR | D3 | D3R | D4, [...]
```

DUT topology table

Clicking a button in the Add Structure section adds the corresponding structure to the DUT model table.



Remote command:

```
[SENSe:]SNPMeasure:DUT:PART<pos>?
[SENSe:]SNPMeasure:DUT?
```

Left Side Label/Right Side Label ← DUT topology table

The DUT is modeled with a left and a right side. Entering a descriptive text for each side facilitates identifying a port during configuration and measurement.

Remote command:

```
[SENSe:]SNPMeasure:DUT:PREFix
```

Wiring ← DUT topology table

For an endpoint of a through structure, you can optionally define whether it is wired to a transmitter (TX) or receiver (RX). Specifying the wiring for the left side determines the wiring of the right side and vice versa.

For a terminated structure, this element is not available.

Remote command:
`[SENSe:]SNPMeasure:DUT:PART<pos>:WIRE`

Pol. ← DUT topology table
The polarity can only be defined for balanced endpoints of through structures with defined Wiring. For those endpoints, by default the polarity of the upper (physical) port is defined as positive ("+"), the lower as negative ("-"). Toggle the polarity by pressing one of the polarity buttons (➕/➖).

Remote command:
`[SENSe:]SNPMeasure:DUT:PART<pos>:WIRE:POLarity`

Identity ← DUT topology table
The default identity is a number starting from 1 and incrementing with each applicable structure added to the DUT model. You can change the identity to an arbitrary string. The firmware neither requires nor enforces identity uniqueness.

Remote command:
`[SENSe:]SNPMeasure:DUT:PART<pos>:WIRE:IDENtity`

Port/Label ← DUT topology table
Physical DUT ports must be numbered consecutively. For identification purposes, you can assign a label to each physical DUT port. The default label is blank.

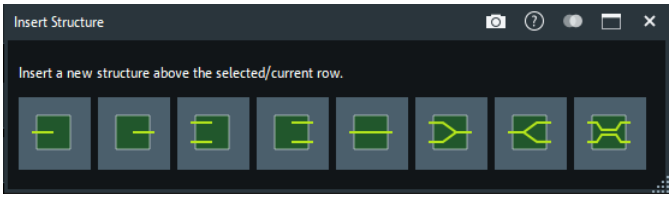
Remote command:
`[SENSe:]SNPMeasure:DUT:PART<pos>:PORT`

Modeling helper controls
On the right side of the "DUT Topology" dialog, there are various controls that can help you to set up and modify the structure of your DUT.

They are arranged in five sections: Structure, Auto Number Ports, Use Template, VNA, and Reference Impedance.

Structure ← Modeling helper controls
The controls in the "Structure" section implement straight-forward operations on the DUT topology table.

- **"Insert"** opens a dialog that lets you insert a structure below the currently selected one.



You can select the same structures as for Add Structure.

"Delete"/"Delete All" deletes the currently selected structure or all structures, respectively.

"Move Up"/"Move Down" moves the currently selected structure one position up/ down in the DUT topology table, without changing the port numbering or other properties of the affected structures.

"Copy Left Label" enabled (default): Whenever you modify a port label on the left side of a structure, the label is automatically copied to the corresponding right side of the structure, overwriting the existing label. For mixed-mode through structures, copying applies to the first port on the left and right side.

Remote command:
`[SENSe:]SNPMeasure:DUT:PART:ADD`
`[SENSe:]SNPMeasure:DUT:PART<pos>:MOVE`
`[SENSe:]SNPMeasure:DUT:PART<pos>:DELete`
`[SENSe:]SNPMeasure:RST`
(Label: `[SENSe:]SNPMeasure:DUT:PART<pos>:PORT`)

Auto Number Ports
Applies an automic numbering to the physical DUT ports. Previously assigned port numbers are overwritten.

With L and R denoting the number of (physical) ports on the left and right side of the DUT:

- "Odd on Left" assigns numbers 2l-1 (l=1,...,L) to the ports on the left, and numbers 2r (r=1,...,R) to the ports on the right (from top to bottom).
- "Sequential" assigns port numbers 1 to L to the ports on the left, and port numbers L+1 to L+R to the ports on the right (from top to bottom).

Note that "Odd on Left" produces an invalid numbering, if |L-R| > 1.

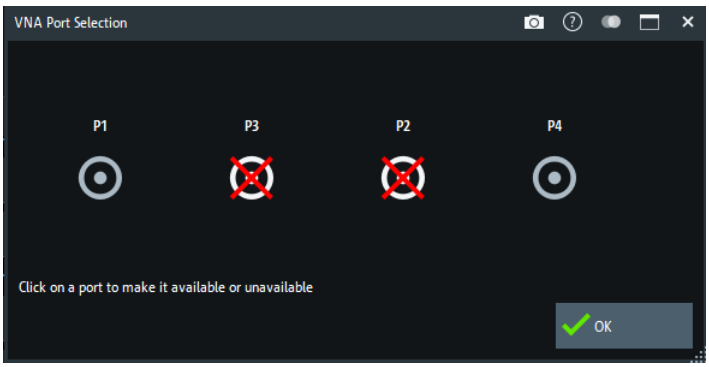
Remote command:
`[SENSe:]SNPMeasure:DUT:NUMBer`

Use Template
The SNPA comes with a set of pre-configured DUT models. These templates pre-populate the DUT topology table, with default structures, wiring, numbering and labeling. You can then modify the model according to your actual setup.

Setting the template to "—NONE—" clears the DUT topology table (same as "Delete All").

Remote command:
`[SENSe:]SNPMeasure:DUT:TEMPlate`

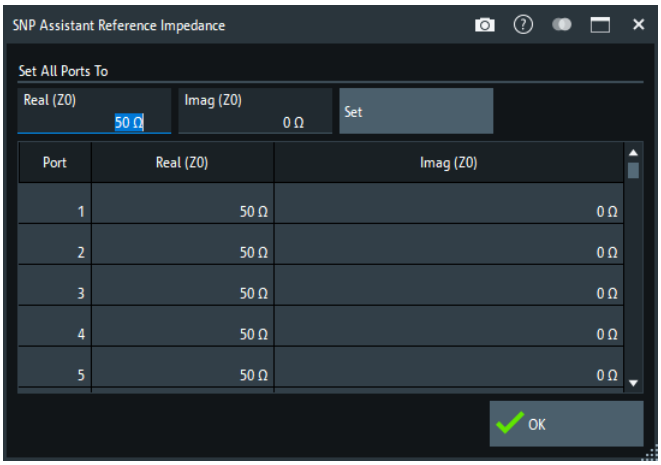
VNA
Use "Select Ports..." to limit the ports on the VNA and connected switch matrices that are available for measuring the modeled DUT.



By default, the SNPA uses **all** available ports – disregarding their state in the logical port configuration (recommended).

Remote command:
`[SENSe:]SNPMeasure:INSTrument:PORT`

Reference Impedance
The "Config..." button brings up a dialog that allows you to specify the reference impedances of the physical DUT ports. The default impedance is 50 Ω.



Use the controls in the "Set All Ports To" section to set a common reference impedance for all ports.

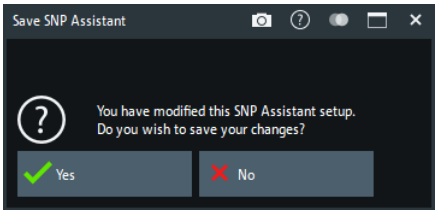
Remote command:
`[SENSe:]SNPMeasure:DUT:RIMPedance`

Assistant controls
Use the **"Next"** button to proceed to the SNP Assistant Overview.

The **"Save..."** button opens a standard dialog that allows you to persist the current setup (topology and data) to a `*.bvm` file.

This file format is portable between all Rohde & Schwarz network analyzers supporting the SNPA feature (option K100). However, on a R&S ZNA it requires firmware version 3.0, and on a R&S ZNB3000, R&S ZNB, R&S ZNBT or R&S ZND it requires firmware version 4.0.

Use **"Close"** to leave the SNPA. The assistant prompts you to persist unsaved changes (~"Save...").

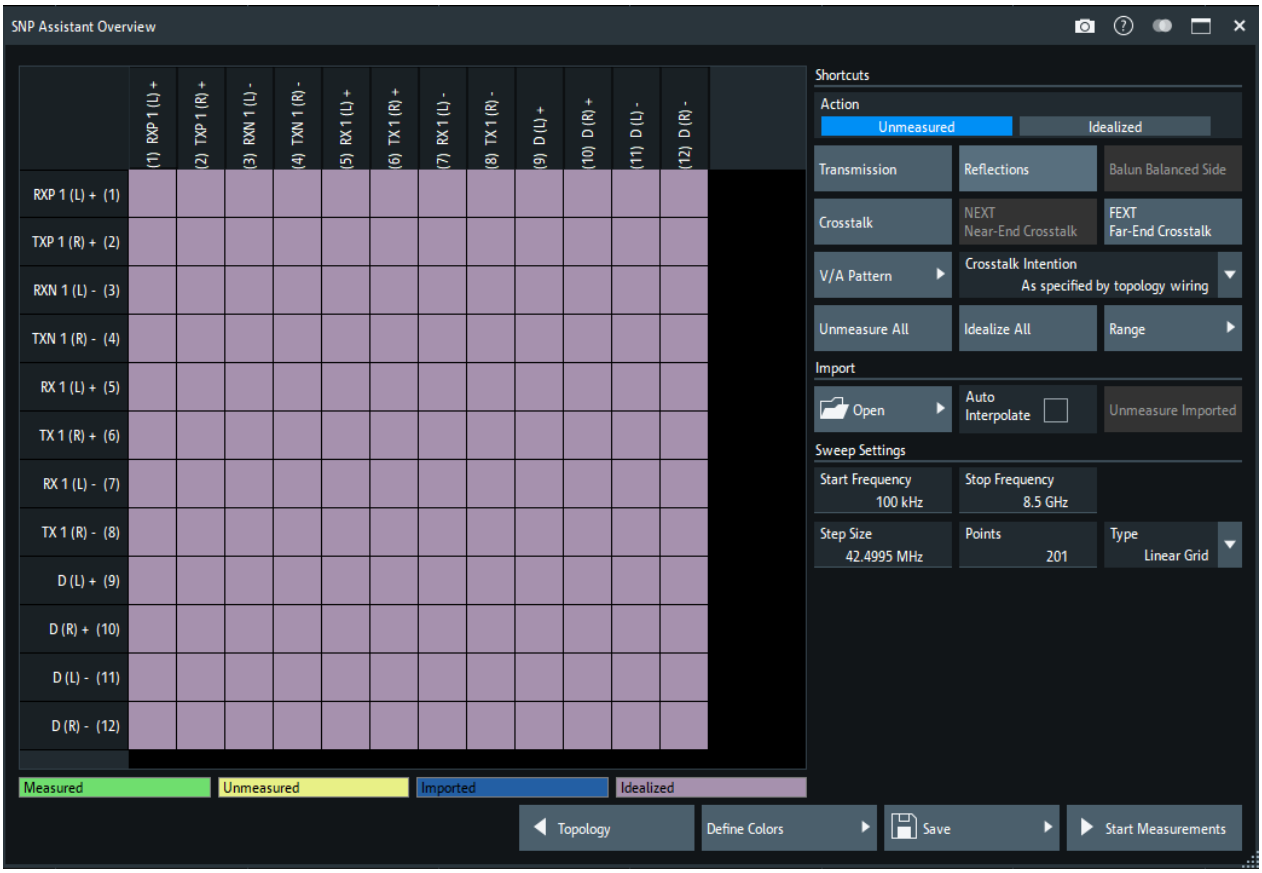


Remote command:
`[SENSe:]SNPMeasure:SAVE`

SNP Assistant Overview

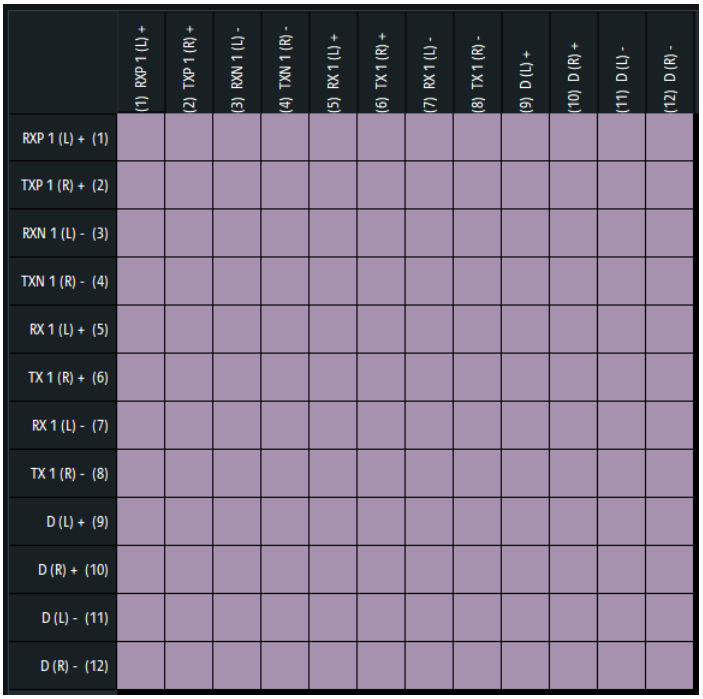
In the "SNP Assistant Overview" you can:

- Select the S-parameters to be measured or loaded
- Select the S-parameters to be idealized (w.r.t. their transmission/reflection properties)
- Specify the sweep settings
- Load S-parameters from Touchstone file



S-matrix panel

The quadratic S-matrix panel in the top-left part of the SNPA overview visualizes the data state of the DUT's S-matrix.

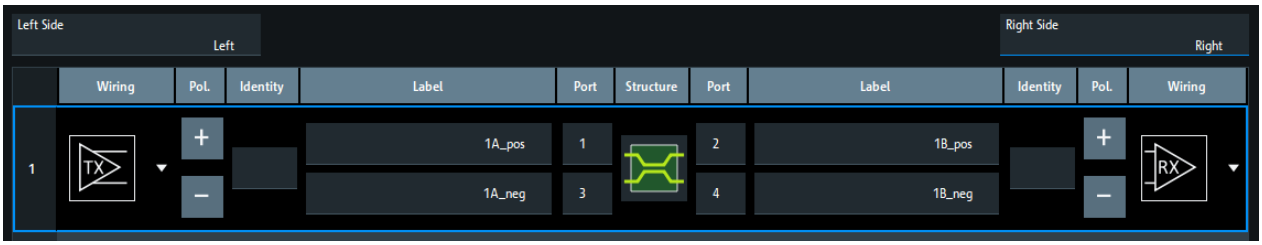


Port labeling ← S-matrix panel

The rows and columns of the S-matrix panel represent the physical DUT ports, ordered by port number. The ports are labeled according to your settings in the topology table:

[<Side Label>] [<Wiring>] [<Identity>] [<Polarity>] [<Label>] (<Port number>)

For example, a DUT structure and labeling like this



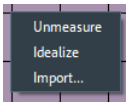
results in a board labeling like that



Data states ← S-matrix panel
Each small square in the S-matrix panel represents a (single-ended) S-parameter, whose data state is indicated by the square's background color (see "Define Colors" on page 425).

- **Imported**: The S-parameter data were imported from Touchstone file
- **Measured**: The S-parameter data were measured, possibly during a previous run of the SNPA (see "SNP Assistant Measurements dock widget" on page 426)
- **Unmeasured**: The S-parameter was marked for measurement, but has not been measured yet.
- **Idealized** (default): Unless marked for measurement, measured or loaded from Touchstone file, an S-parameter is assumed to have ideal properties, according to the DUT topology.

Tapping/clicking a square brings up a context menu that lets you change the data state of the corresponding S-parameter:

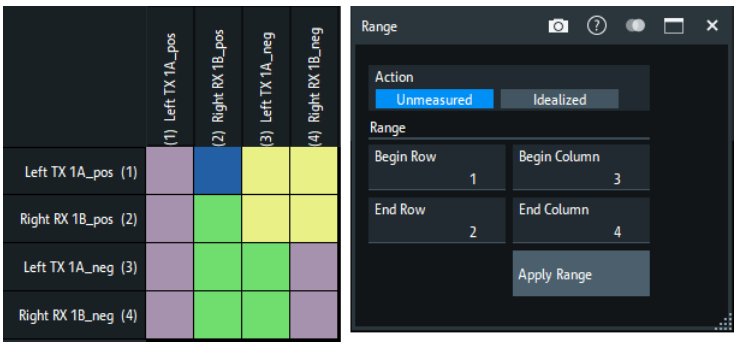


To change the data state of certain ranges or logical groups of S-parameters in one go, use the controls in the S-Parameters or in the Shortcuts section.

S-Parameters
The controls in this and the Shortcuts section allow you to specify the S-parameters to be measured, idealized, or loaded from Touchstone file.

Unmeasured/Idealize ← S-Parameters
Preselects the target state of subsequent state settings, applied in the Range or Shortcuts section.

Range ← S-Parameters
Allows you to select a rectangular range in the S-matrix panel.



With "Set", you can then apply the preselected measurement state to the corresponding S-parameters.

Remote command:
`[SENSe:]SNPMeasure:DUT:INIT:RANGe`

Quick Set ← S-Parameters
"Unmeasure All"/"Idealize All" sets the entire S-matrix to "Unmeasured"/"Idealized" state.

Remote command:
`[SENSe:]SNPMeasure:DUT:INIT:RANGe` on page 1457

Data ← S-Parameters
Select **"Import"** to browse for a Touchstone file to import S-parameter data from. If the integrity check passes, the SNPA prompts you with the "Data Importer" dialog that allows you to map a subset if the Touchstone file's port numbers to the corresponding physical port names of the DUT (as shown on the S-matrix panel).

On the S-parameter board, the imported S-parameters are highlighted in the "Imported" color, which is blue by default (see "Define Colors" on page 425). E.g., if you map DUT ports 1 to 4 in the "Data Importer", then the background of the upper left 4x4 square on the S-parameter board is set to the "Imported" color:

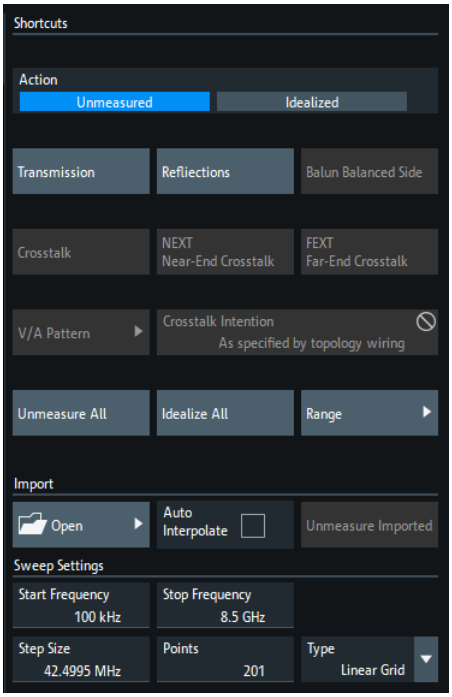
Use "Unmeasure Imported" (see below) to get rid of previously imported data.

Enable **"Auto Interpolate Imported File"** to accept interpolation automatically during the integrity, if this is sufficient to pass the check. Otherwise the "SNP Assistant Integrity Check" pops up every time interpolation is an option (default).

"Unmeasure Imported" removes any imported S-parameter data and sets the state of the corresponding S-parameters to unmeasured.

Remote command:
`[SENSe:]SNPMeasure:DUT:IMPort`

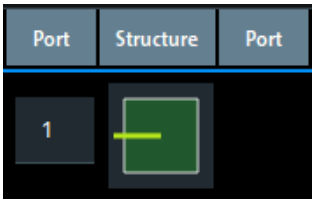
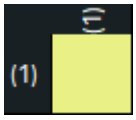
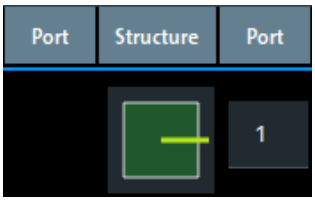
Shortcuts
Applies the selected state to certain logical groups of S-parameters. The application is cumulative, i.e. if you select "Transmission" and "Reflection", then the selected state is applied to both reflection and transmission parameters.

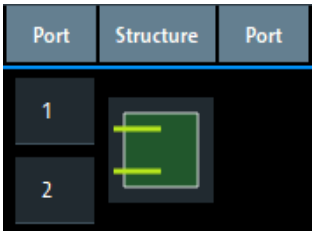
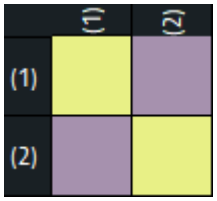
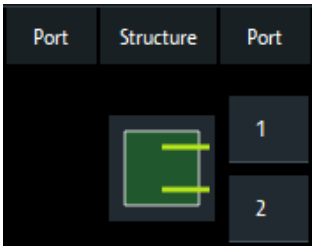
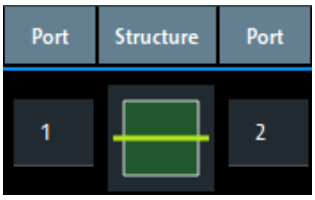
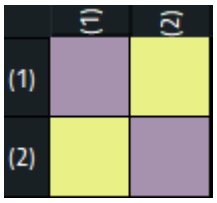
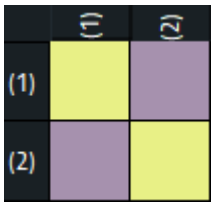
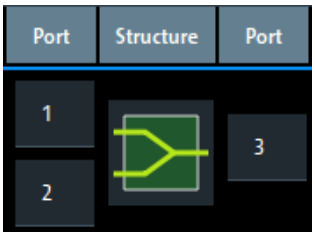
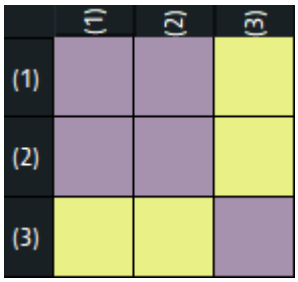
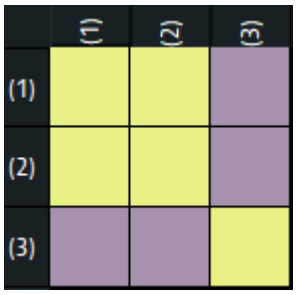
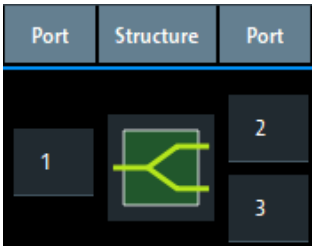
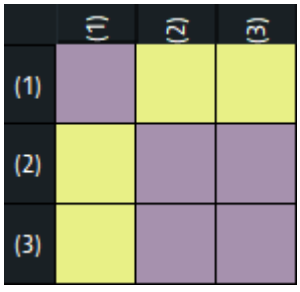
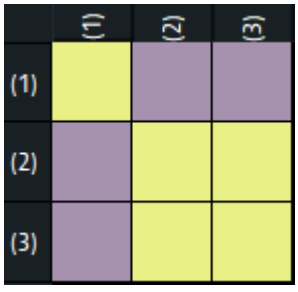


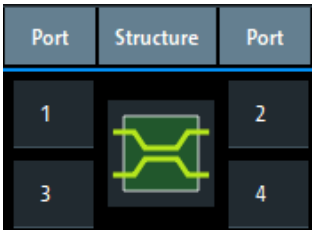
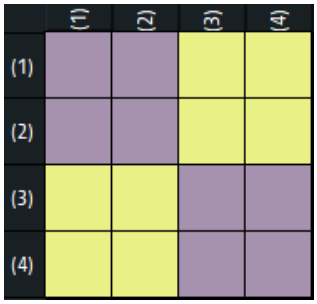
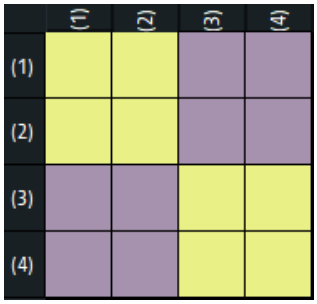
Remote command:
`[SENSe:]SNPMeasure:DUT:INIT:IDEalize`
`[SENSe:]SNPMeasure:DUT:INIT:MEASure`

Transmission/Reflections ← Shortcuts
Refer to transmission and reflection S-parameters **within a DUT structure**, respectively (see "Add Structure" on page 412).

Table 5-4: S-parameter selection using Transmission/Reflection (sequential port numbering)

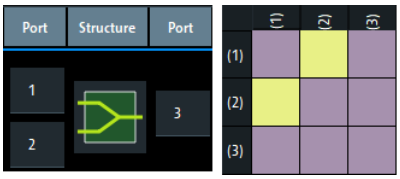
Structure*	"Transmission"	"Reflection"
	n.a.	
		

Structure*	"Transmission"	"Reflection"
	n.a.	
		
		
		
		

Structure*	"Transmission"	"Reflection"
		
* sequential port numbering.		

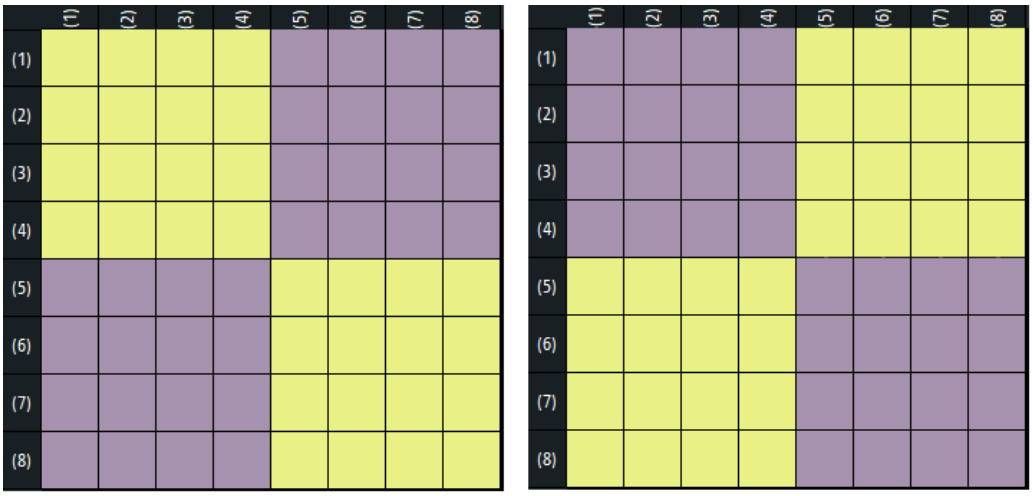
Note that the "Transmission" and "Reflections" patterns depend on the port numbering.

Balun Balanced Side ← Shortcuts
This button is only enabled if you have defined mixed structures in the DUT (balanced-to-single-ended or single-ended-to-balanced). For each of these structures, "Balun Balanced Side" refers to S-parameters between the two physical ports on the balanced side:



Crosstalk ← Shortcuts
Refers to S-parameters that are related to signal propagation between *different* DUT structures. Hence this button is only enabled if you have defined at least two structures.

In the S-matrix panel, "Crosstalk" S-parameters complement the "Transmission" and "Reflection" S-parameters. For example, with two balanced through structures at physical ports 1 to 4 and ports 5 to 9, respectively, we have the following patterns:

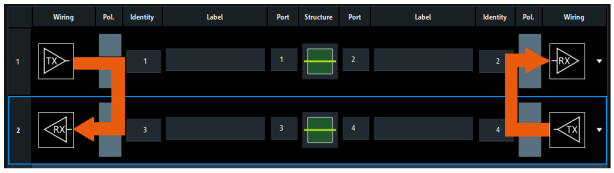


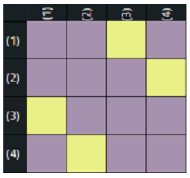
left = "Transmission" + "Reflection" S-parameters
right = "Crosstalk" S-parameters

NEXT (Near-End Crosstalk)/FEXT (Far-End Crosstalk)/Crosstalk Intention ← Shortcuts
Which S-parameters are related to "NEXT (Near-End Crosstalk)", and which ones to "FEXT (Far-End Crosstalk)", depends on your DUT model and the selected **"Crosstalk Intention"**.:

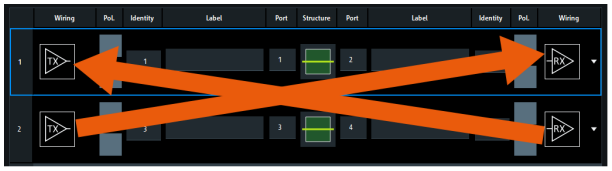
"As specified by topology wiring" (default)

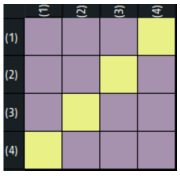
- "NEXT (Near-End Crosstalk)": Propagation to a structure's RX from the TX of a structure with inverse wiring (and vice versa)





- "FEXT (Far-End Crosstalk)": Propagation to a structure's RX from the TX of another structure with the same wiring (and vice versa)





"NEXT (Near-End Crosstalk)" requires two DUT structures with inverse wiring (TX of structure 1 and RX of structure 2 on the same side of the DUT).
"FEXT (Far-End Crosstalk)" requires two DUT structures with same wiring (TX of structure 1 and RX of structure 2 on opposite sides of the DUT).

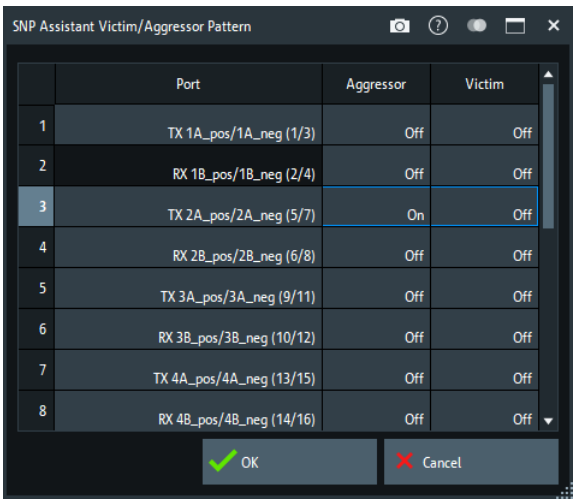
"As specified by V/A pattern"

- "NEXT (Near-End Crosstalk)": Propagation to a victim from an aggressor on the same side of the DUT (and vice versa)
- "FEXT (Far-End Crosstalk)": Propagation to a victim from an aggressor on the opposite side of the DUT (and vice versa)

Both shortcuts only take effect if you specify a non-empty V/A pattern.

"RX (<port number(s)>)" Similar to "As specified by topology wiring", but limited to the selected receiver

V/A Pattern ← Shortcuts
"V/A Pattern" brings up a dialog that allows you to specify the physical DUT ports to be considered as aggressors or/and victims:



The V/A pattern above represents the crosstalk intention "As specified by topology wiring".

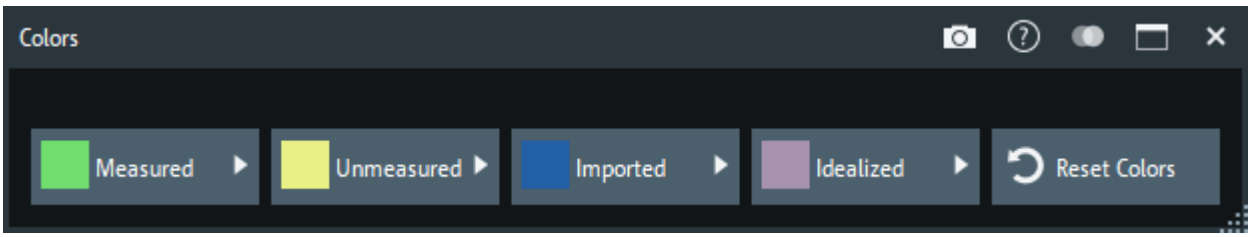
Sweep Settings
The sweep settings of the current SNPA project. For a description of the parameters, see Section 5.10, "Sweep Softtool", on page 536.



When you start a new project, the SNPA uses the sweep settings of the active channel. Changing the sweep settings in the SNPA immediately applies to the active channel. Similarly, if you recall an existing project, the SNPA applies the project's sweep settings to the active channel.

The parameters are editable until S-parameter data is added to the SNPA project, either by measurement of import. To make them editable again, all measured and imported S-parameters must be either idealized or unmeasured.

Define Colors
Opens a dialog that allows you to define the state-dependent coloring of the S-matrix panel.



The SNPA state color scheme is a global anlyzer setting, and is **not** stored in the SNPA project.

"Reset Colors" reverts the scheme to its default colors.

Remote command:
[SENSe:]SNPMeasure:DISPlay:CMAP<pos>:RGB on page 1455
[SENSe:]SNPMeasure:DISPlay:RST on page 1456

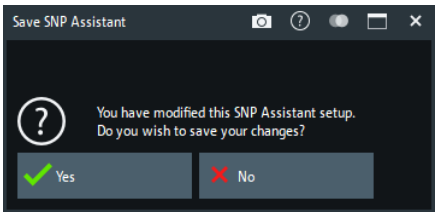
Assistant controls
Use the **"Previous"** button to go back to the SNP Assistant DUT Topology view.

Use the **"Start"** button to proceed to the SNP Assistant Measurements dock widget.

The **"Save..."** button opens a standard dialog that allows you to persist the current setup (topology and data) to a `*.bvm` file.

This file format is portable between all Rohde & Schwarz network analyzers supporting the SNPA feature (option K100). However, on a R&S ZNA it requires firmware version 3.0, and on a R&S ZNB3000, R&S ZNB, R&S ZNBT or R&S ZND it requires firmware version 4.0.

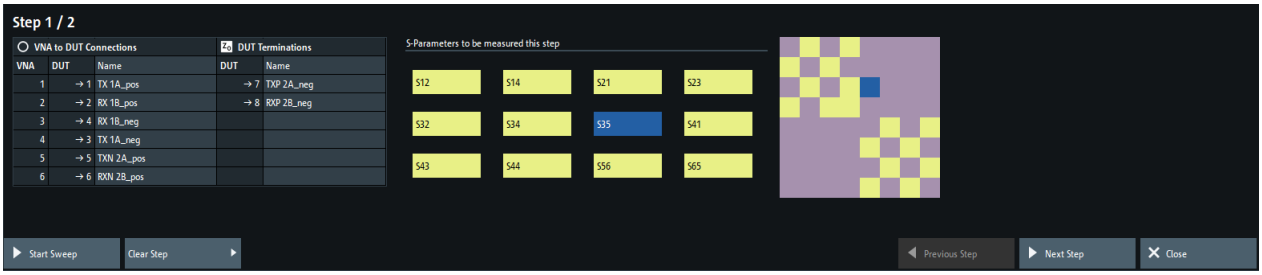
Use **"Close"** to leave the SNPA. The assistant prompts you to persist unsaved changes (~"Save...").



Remote command:
[SENSe:]SNPMeasure:SAVE

SNP Assistant Measurements dock widget

The "SNP Assistant Measurements" dock widget guides you through the steps needed to measure the modeled DUT.



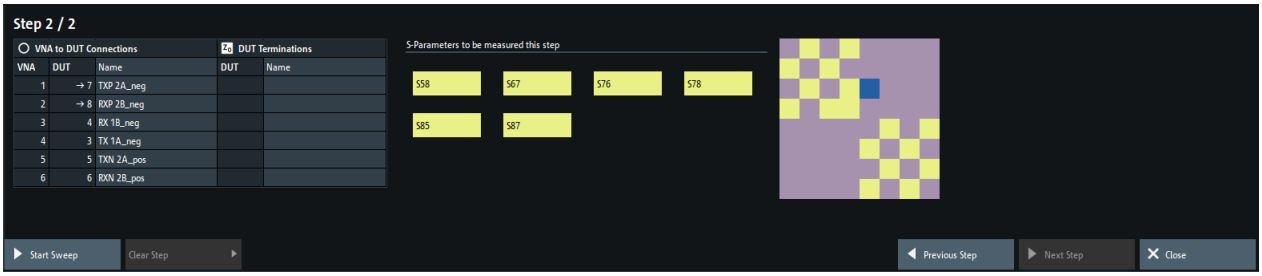
Depending on the number of ports on the VNA (and connected switch matrices), the number of DUT ports, and the S-parameters you do not assume "ideal", the SNPA algorithm determines the required measurement steps. The steps are optimized w.r.t. the number of cable changes.

The dock is divided into three sections. The table on the left indicates the VNA to DUT Connections and DUT Terminations required for the current measurement step. On the right, there is a scaled down version of the S-matrix panel that shows the data states of the DUT's S-parameters. The buttons in the middle represent the S-parameters to be measured in this step (along with their data states) and allow you to import the respective S-parameter data from Touchstone file.

In the example above, we have a 4-port VNA and a DUT with two balanced through structures (DUT ports 1 to 4 and 5 to 8). The data of the second structure were already imported from Touchstone file (blue color in the original color scheme. The crosstalk S-parameters were idealized (no crosstalk; purple color in the original color scheme). Only the S-matrix of the first structure was marked unmeasured (yellow color in the original color scheme).

The SNPA then mandates two measurement steps:

- Step 1 allows you to measure structure 1, with the 4 VNA ports connected to the 4 ports of this structure (Start Sweep.
- In step 2, structure 2 is presented – even though the related S-parameter data were already imported and hence the step is marked as "Completed". To overwrite existing S-parameter data, you can use the Clear Step button to clear previously measured and/or imported data for this step. You can then renew the data imports (see S-parameters: button grid and panel) and/or repeat the measurements (Start Sweep).



After completing all measurement steps, you can use the Export snp File... on the "SNP Assistant" softtool tab to export the resulting 8x8 S-matrix to an `s8p` Touchstone file.

Step heading

The SNPA algorithm determines the required measurement steps. The heading of the dock widget indicates the current step number, the total number of steps, and the step completeness.

Step 1 / 2 **Step 2 / 2 (Completed)**

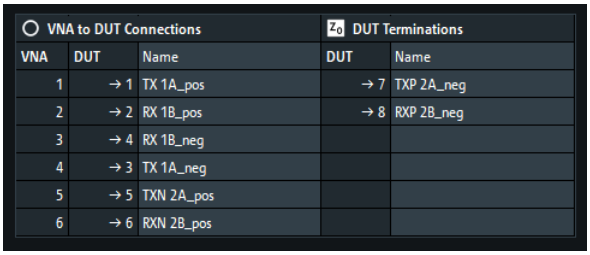
A step is considered "Completed", if all its related S-parameters are in data state "Measured" or "Imported".

When all steps are completed, you can use the Export snp File... button on the "SNP Assistant" tab to export the DUT's full S-matrix to a Touchstone file.

Remote command:
`[SENSe:]SNPMeasure:STATus?`

VNA to DUT Connections and DUT Terminations

This table displays the "VNA to DUT Connections" and "DUT Terminations" you have to establish for the current measurement step.

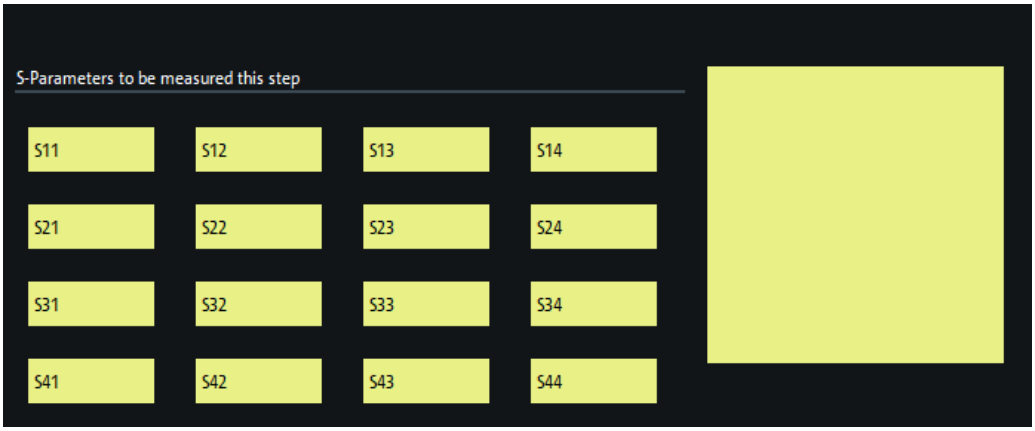


Necessary cable and termination changes are indicated in red, with an arrow symbol (→) next to the DUT port number.

The names of the ports follow the labeling scheme of the S-matrix panel in the SNP Assistant Overview.

Remote command:
`[SENSe:]SNPMeasure:STEP<pos>?`

S-parameters: button grid and panel
The buttons in the middle of the dock widget represent the S-parameters to be measured in the corresponding step (along with their data states). The panel on the right is a scaled down version of the S-matrix panel that shows the data states of the DUT's S-parameters:



Tap or click one of the buttons to bring up a file browser for Touchstone files, which allows you to import or re-import the respective S-parameter data from file.

Remote command:
`[SENSe:]SNPMeasure:STEP<pos>?`
`[SENSe:]SNPMeasure:STEP<pos>:IMPort`

Start Sweep
Once you have established the requested VNA to DUT Connections and DUT Terminations, you can use the "Start Sweep" button to measure the "Unmeasured" S-parameters of the current SNPA step.

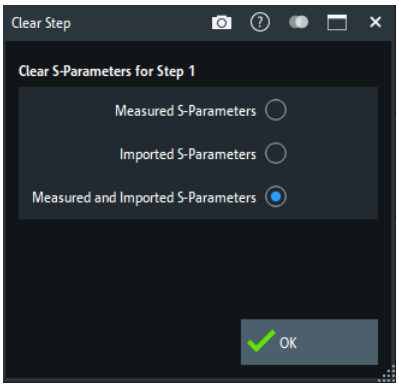
Note
- If the SNPA project already contains measured or imported data, an integrity check is performed (see "Measurement" on page 430).

- If none of the step's S-parameters is in "Unmeasured" state, the "Start Sweep" button is disabled. Use the Clear Step button if you want to repeat some of the measurements, or replace imported by measured data.

Remote command:
`[SENSe:]SNPMeasure:STEP<pos>:MEASure`

Clear Step
If none of the S-parameters related to a measurement step is in "Unmeasured" state, the "Start Sweep" button is disabled. Use the "Clear Step" button if you want to repeat some of the measurements, or replace imported by measured data.

If you click the button, the SNPA brings up a dialog that allows you to select the S-parameters to "unmeasure".



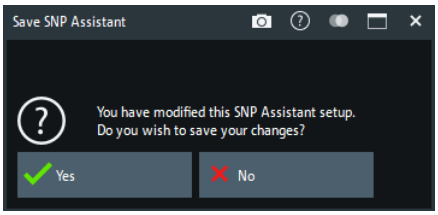
You can then either measure these S-parameters, or import their traces using the buttons in the step's button grid.

Remote command:
`[SENSe:]SNPMeasure:STEP<pos>:CLEar`

Other assistant controls
Use the "Previous" or "Next" button to go back to the previous measurement step or proceed to the next one, respectively.

Save... is now available on the softtool tab.

Use **"Close"** to leave the SNPA. The assistant prompts you to persist unsaved changes (~"Save...").



Note: An SNPA project not only comprises the DUT setup (topology, port numbering and labeling) and the measurement setup (scope, sweep settings, data state), but also the available data – no matter if measured or loaded from Touchstone file. So if you want to persist the measured and imported S-parameter data for later use within the SNPA, you have to save the project after the measurement.

Integrity check

The "SNP Assistant Integrity Check" ensures consistent S-parameter data within the SNPA project. Currently, the check focuses on identical – or at least compatible – sweep settings.

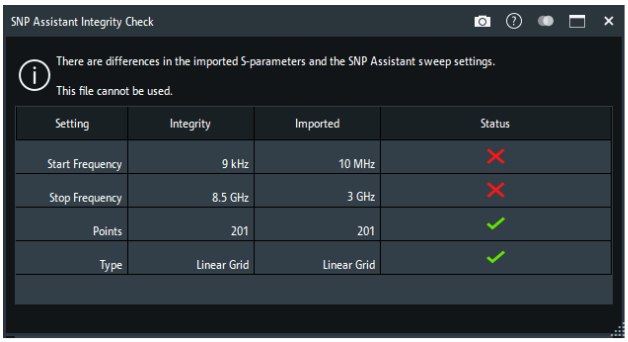
As long as the project does not contain S-parameter data, i.e. if none of the project's S-parameters is in "Measured" or "Imported" state, you can change the Sweep Settings as you like. With the first import or measurement, however, the project's sweep settings get locked, and subsequent import or measurement attempts are checked against these sweep settings (**"Integrity"**).

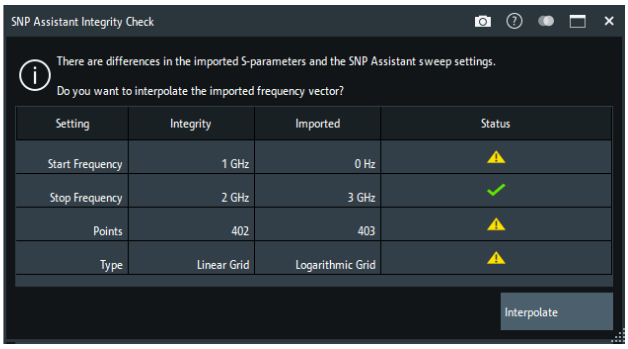
To clear the SNPA project from S-parameter data and disable the integrity check, set all "Measured" or "Imported" S-parameters to "Unmeasured" or "Idealized".

Import
When you attempt to import S-parameters, the SNPA checks whether the frequency points of the loaded Touchstone file are compatible with the project's Sweep Settings.

Depending on the detected differences, the SNPA either rejects the import, or asks you for confirmation on:

- Adjusting the sweep settings ("Use Imported Settings")
  Choose this option to adjust and lock the SNPA's sweep settings to the sweep settings of the Touchstone file, and to import the selected S-parameters without interpolation.
- Performing data interpolation ("Interpolate")
  Choose this option to keep SNPA's sweep settings (and lock them), and to import the selected S-parameters with interpolation.





Remote command:
n.a.
Via remote control, interpolation of imported data is always accepted, if possible. Other incompatibilities are always rejected.

Measurement
When you attempt to measure S-parameter data (Start Sweep), the SNPA checks whether the active channel's sweep settings match the current project's Sweep Settings.

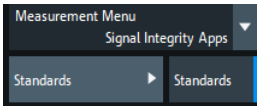
If the check fails, you can either adjust the channel's sweep settings to the project's sweep settings and continue with the measurement sweep ("Restore Integrity"), or "Cancel" the measurement sweep.

Remote command:

```
[SENSe:]SNPMeasure:MINTegrity
[SENSe:]SNPMeasure:MINTegrity:RESTore
```

5.2.7.2 Standards tab

The "Standards" tab allows you to set up your VNA for signal integrity tests according to various industry standards.

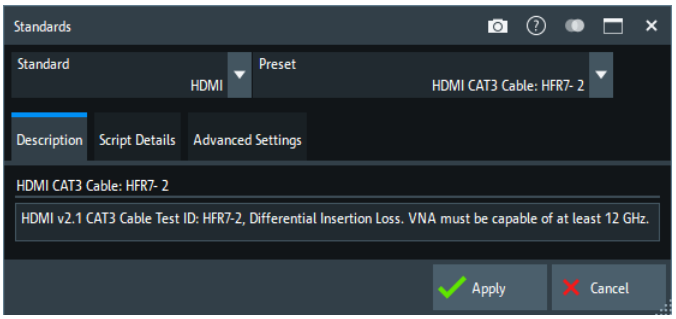


Use the "Standards" button to open the Standards dialog offering predefined SCPI-based instrument setup scripts.

Standards dialog

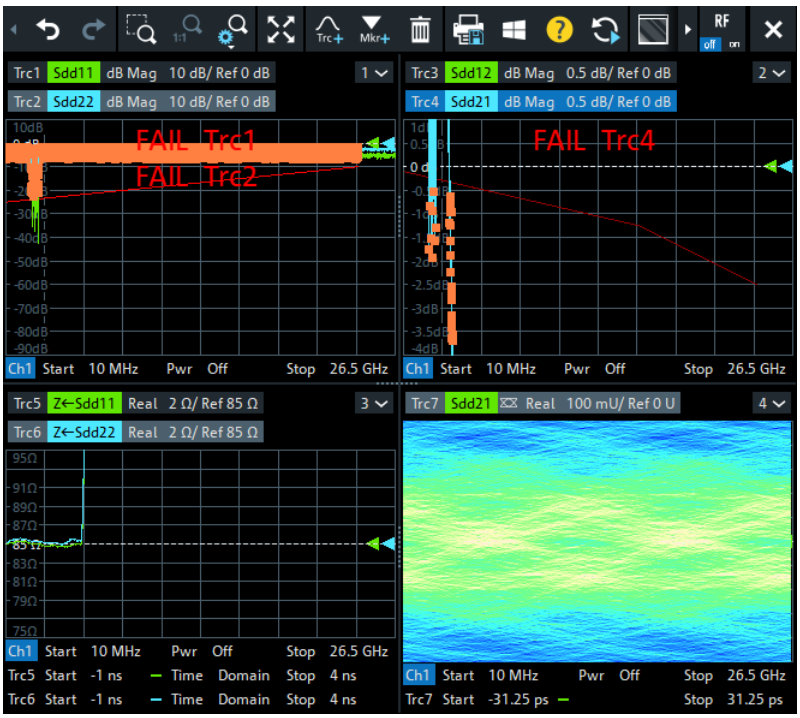
Allows you to set up your instrument for measuring DUTs of various industry standards.

The setups are based on predefined SCPI-based "Preset" scripts with IECWin special commands, which enable conditional script execution based on the properties of your instrument (frequency range, number of ports, available options, ...).



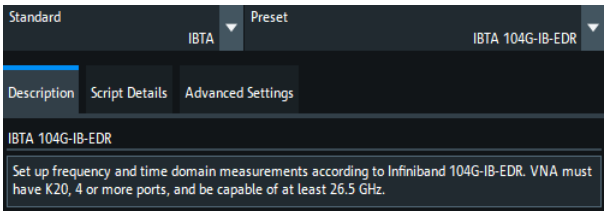
Access: Standards tab > "Standards"

If you execute the "PCIe 5 Connector" script, for example, the result looks something like this:

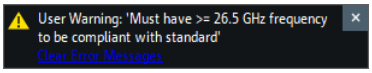


Description tab

Provides a description of the selected "Preset" script, specifying the requirements for successful execution (number of ports, frequency range, available options, ...).

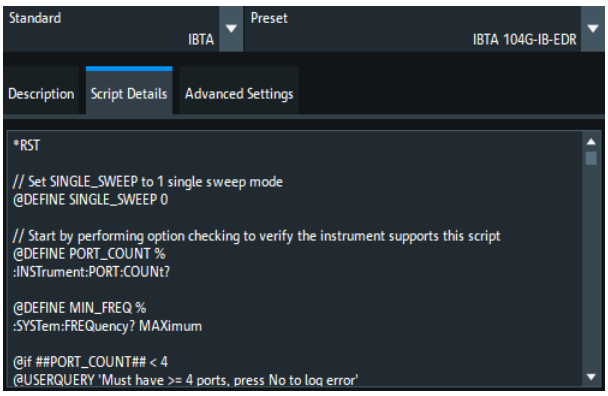


The predefined scripts do not stop on minor errors, such as an insufficient maximum frequency. However, in this case they tell you that the resulting setup and measurements do not meet the requirements of the underlying standard conformance tests.



Script Details tab

Mirrors the SCPI/IECWin commands to be executed for the selected "Preset".

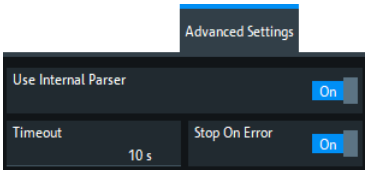


In the screenshot above, you can see the script using IECWin variable definitions (`@DEFINE ...`), conditional command execution (@if <condition> ... @else ...) and user interaction (`@USERQUERY ...`).

Advanced Settings tab

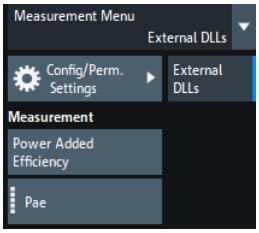
Allows you to configure how the script is executed:

- Use the internal parser (default) or IECWin (a.k.a. "GPIB Explorer")
- Timeout
- Stop or continue on error



5.2.8 External DLLs

Measurement Add-Ins

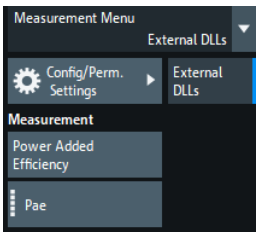


5.2.8.1 External DLL tab

The "External DLL" tab allows you to add/remove custom plug-ins to/from the firmware. It gives access to their configuration, tasks, and traces.

Background information
Refer to Section 4.7.31, "External DLLs", on page 282.

Controls on the External DLL tab



Configuration / Perm. Settings
Opens the External DLL Configuration dialog.

Measurement
The "Measurement" section gives access to the custom traces of the loaded plug-ins.

Each external DLL `<ext_dll_name>.dll` that provides at least one custom trace type, adds a measurement button "<ext_dll_name>" (with dotted left border) to the "Measurement" section of the "External DLL" tab. If neither the PAE plugin nor any custom external DLL is installed, the "Measurement" section is hidden.

Tapping (clicking) or dragging & dropping a measurement button to the diagram area, opens an additional Trace Definition dialog.

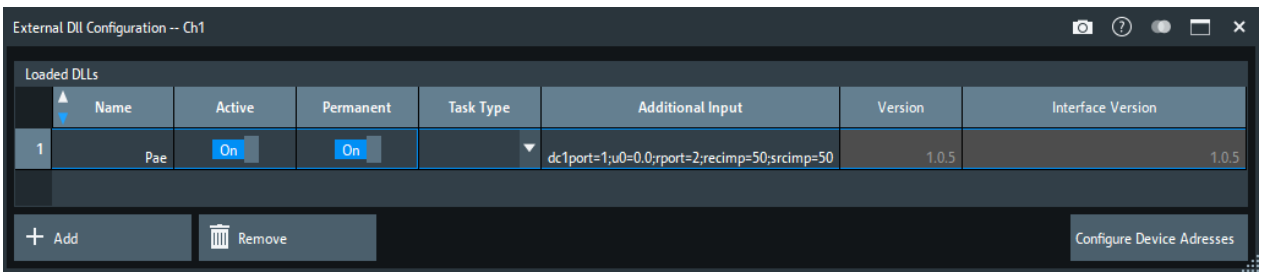
As usual, tapping (clicking) redefines the current trace while dragging & dropping creates a trace.

External DLL Configuration dialog

In the "External DLL Configuration" dialog, you can add/remove External DLLs to/from the firmware. If a plugin requires interaction with an external device, you can define its address from here.

To add/remove external DLLs, the firmware has to be run as administrator.

Close the analyzer GUI, tap and hold (right-click) the VNA's shortcut icon on the desktop and select "Run as administrator".



Loaded DLLs table
Displays the loaded plug-in DLLs and their configuration in the current channel.

Remote command:
[SENSe:]CDLL:LIST?

Active ← Loaded DLLs table
Allows you to de-/activate the respective plugin in the current channel. An active plugin can be used to create traces and/or can be made Permanent.

Remote command:
[SENSe<Ch>:]CDLL[:STATe]

Permanent ← Loaded DLLs table
Defines the task that is selected via Task Type and further defined via "Additional Input" on page 435 as permanent.

A permanent task can interact with the firmware, even if it currently does not provide any traces, e.g. for synchronized control of an external device. In contrast to the trace mode, the permanent mode of operation is not restricted to a particular driving port.

Remote command:
[SENSe<Ch>:]CDLL:PERManent[:STATe]

Task Type ← Permanent ← Loaded DLLs table
Displays the task types implemented by the external DLL. This combo-box is only enabled, if the DLL is marked as permanent.

Remote command:
[SENSe<Ch>:]CDLL:PERManent:TASK

Additional Input ← Permanent ← Loaded DLLs table
Specifies additional input that further specifies the permanent task to be performed by the external DLL.

This string is processed by the external DLL, with DLL-specific syntax and semantics. If it is malformed or inappropriate for some reason, the firmware logs and displays the error messages that are returned by the DLL. See the docs of your DLL for details.

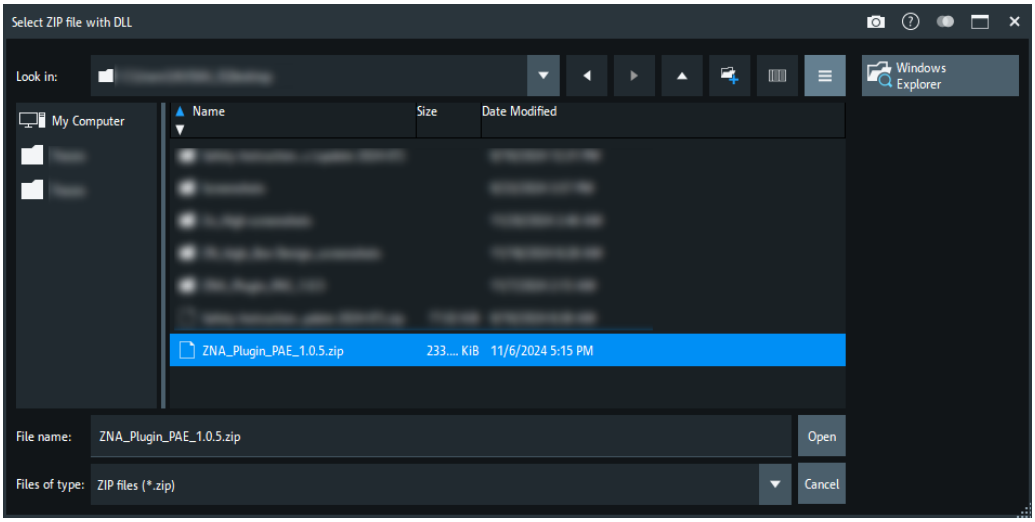
Remote command:
[SENSe<Ch>:]CDLL:PERManent:ADDitional

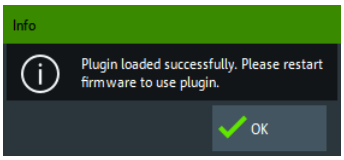
Version/Interface Version ← Loaded DLLs table
"Version" is purely informative. It reports the version of the DLL, as specified by its developer.

"Interface Version" is the version of the plug-in SDK that was used to compile the DLL (more precisely, the version of the interface `RsVnaCustomDllInterface.h` against which it was compiled). This version can change the DLL handling of the analyzer firmware.

Add / Remove
Allows you to add/remove custom plugins to/from the analyzer firmware.

A plug-in is installed as a `*.zip` file, containing a DLL and, possibly, some additional files.



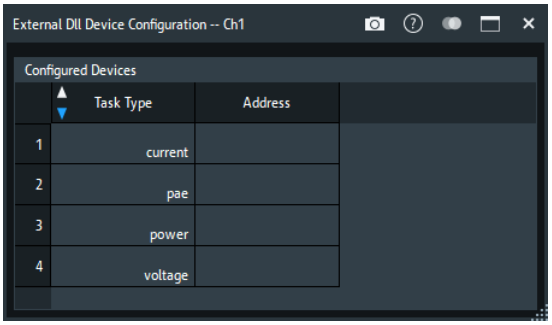


The firmware must be run as administrator to add or remove custom plugins. Once installed, standard user privileges are sufficient to use a plugin's functionality.

Remote command:
`[SENSe:]CDLL:ADD`
`[SENSe:]CDLL:REMove`

Configure Device Adress
Opens an additional dialog that allows you to specify the addresses of external devices the external DLL requires to complete its tasks. For each task type, a single device address can be configured.

The address syntax is DLL-specific: the DLL manages the connection to the external device. In the example below, the DLL expects a VISA resource string and the VNA operator indicates that the device is connected to (virtual) COM port 4.



Remote command:
`[SENSe<Ch>:]CDLL:TASK:ADDRess`

Trace Definition dialog

Specifies a custom trace that is provided by an external DLL.

Access: External DLL tab > "Measurement" section buttons

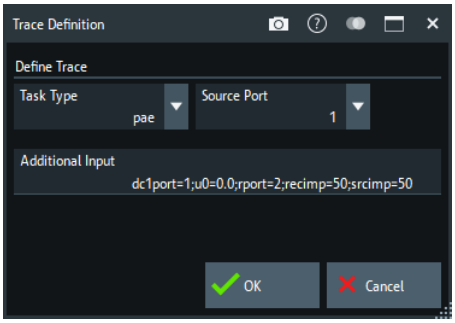


Figure 5-21: Trace Definition dialog

In the VNA firmware, the following string represents the measured quantity:

Cu(P<Src Port>)<lowercase_ext_dll_name>/<Task Type>/<Additional Input>

This string is displayed in the trace info and in the "Trace Manager". You can also use it to specify the corresponding <Result> in remote commands:

- `CALCulate<Ch>:PARameter:MEASure`
- `CALCulate<Ch>:PARameter:SDEFine`

Define Trace
Requests the external DLL, for which the Trace Definition dialog was called, to calculate a particular trace.

- "Task Type" selects the task that is responsible for calculating this kind of trace
- The trace is calculated when "Src Port" is the (physical) driving port

In the Figure 5-21, for example, the external DLL provides a task to calculate PAE (power added efficiency) traces. The user requests the PAE trace calculation while port 1 is the driving port.

"Task Type" and "Src Port" define the first part of the trace identifier:

Cu(P<Src Port>)<lowercase_ext_dll_name>/<Task Type>/<Additional Input>

If these selections are insufficient to define the trace completely, Additional Input can be provided to the respective task.

Additional Input
For some task types, the basic information in "Define Trace" can be supplemented with "Additional Input". This string is processed by the external DLL, with DLL-specific syntax and semantics. If it is malformed or inappropriate for some reason, the firmware logs and displays the error messages that are returned by the DLL. See the docs of your DLL for details.

In Figure 5-21, for example, the external DLL plugin is provided with additional information about the source and receive impedance (*recimp=50;srcimp=50*) to calculate the PAE trace correctly.

The "Additional Input" is the last part of the trace identifier:

Cu(P<Src Port>)<lowercase_ext_dll_name>/<Task Type>/**<Additional Input>**

5.3 Format softtool

The "Format" softtool allows you to define how the measured data is presented in the diagram area.

Access: Trace – [Format]

Measured quantities and display formats

The analyzer allows arbitrary combinations of display formats and measured quantities (see Section 5.2, "Meas softtool", on page 291). Nevertheless, to extract useful information from the data, it is important to select a display format which is appropriate to the analysis of a particular measured quantity.

An extended range of formats is available for markers. To convert any point on a trace, create a marker and select the appropriate marker format (see "Marker Format" on page 507). Marker and trace formats can be applied independently.

Background information

Refer to the following sections:

- Section 4.2.3, "Trace formats", on page 108
- Section 4.2.3.3, "Measured quantities and trace formats", on page 116



dB Mag

Selects a Cartesian diagram with a dB scale of the vertical axis to display the magnitude of the complex measured quantity.

Properties: The stimulus variable appears on the horizontal axis, scaled linearly. The magnitude of the complex quantity C, i.e. |C| = sqrt ($Re(C)^2$ + $Im(C)^2$), appears on the vertical axis, scaled in dB. The decibel conversion is calculated according to dB Mag(C) = 20 * log(|C|) dB.

Application: dB Mag is the default format for the complex, dimensionless S-parameters. The dB-scale is the natural scale for measurements related to power ratios (insertion loss, gain etc.).

Tip (alternative formats): The magnitude of each complex quantity can be displayed on a linear scale. It is possible to view the real and imaginary parts instead of the magnitude and phase. Both the magnitude and phase are displayed in the polar diagram.

Remote command:
`CALCulate<Chn>:FORMat MLOGarithmic`

Phase
Selects a Cartesian diagram with a linear vertical axis to display the phase of a complex measured quantity in the range between –180 degrees and +180 degrees.

Properties: The stimulus variable appears on the horizontal axis, scaled linearly. The phase of the complex quantity C, i.e. φ (C) = arctan (Im(C) / Re(C)), appears on the vertical axis. φ (C) is measured relative to the phase at the start of the sweep (reference phase = 0°). If φ (C) exceeds +180° the curve jumps by –360°; if it falls below –180°, the trace jumps by +360°. The result is a trace with a typical sawtooth shape. The alternative "Unwr Phase" format avoids this behavior.

Application: Phase measurements, e.g. phase distortion, deviation from linearity.

Tip (alternative formats): The magnitude of each complex quantity can be displayed on a linear scale or on a logarithmic scale. It is possible to view the real and imaginary parts instead of the magnitude and phase. Both the magnitude and phase are displayed in the polar diagram. As an alternative to direct phase measurements, the analyzer provides the derivative of the phase response for a frequency sweep (Delay).

Remote command:
`CALCulate<Chn>:FORMat PHASe`

Smith
Selects a Smith chart to display a complex quantity, primarily a reflection S-parameter.

Properties: The Smith chart is a circular diagram obtained by mapping the positive complex semi-plane into a unit circle. Points with the same resistance are located on circles, points with the same reactance produce arcs. If the measured quantity is a complex reflection coefficient S_{ii}, then the unit Smith chart represents the normalized impedance. In contrast to the polar diagram, the scaling of the diagram is not linear.

Application: Reflection measurements; see example in "Smith" on page 111.

Tip: The axis for the sweep variable is lost in Smith charts but the marker functions easily provide the stimulus value of any measurement point. dB values for the magnitude and other conversions can be obtained by the "Marker Format" functions.

Remote command:
`CALCulate<Chn>:FORMat SMITh`

Polar
Selects a polar diagram to display a complex quantity, primarily an S-parameter or ratio.

Properties: The polar diagram shows the measured data (response values) in the complex plane with a horizontal real axis and a vertical imaginary axis. The magnitude of a complex value is determined by its distance from the center, its phase is given by the angle from the positive horizontal axis. In contrast to the Smith chart, the scaling of the axes is linear.

Application: Reflection or transmission measurements, see example in "Polar" on page 110.

Tip: The axis for the sweep variable is lost in polar diagrams but the marker functions easily provide the stimulus value of any measurement point. dB values for the magnitude and other conversions can be obtained by the "Marker Format" functions.

Remote command:
`CALCulate<Chn>:FORMat POLar`

SWR
Calculates the standing wave ratio (SWR) from the measured quantity (typically a reflection S-parameter) and displays it in a Cartesian diagram.

Properties: The SWR (or voltage standing wave ratio, VSWR) is a measure of the power reflected at the input of the DUT. It is calculated from the magnitude of the reflection coefficients S_{ii} (where *i* denotes the port number of the DUT) according to:

$$SWR = \frac{1+|S_{ii}|}{1-|S_{ii}|}$$

The superposition of incident and reflected wave on the transmission line connecting the analyzer and the DUT causes an interference pattern with variable envelope voltage. The SWR is the ratio of the maximum voltage to the minimum envelope voltage along the line.

Interpretation of the SWR

The superposition of the incident wave I and the reflected wave R on the transmission line connecting the analyzer and the DUT causes an interference pattern with variable envelope voltage. The SWR is the ratio of the maximum voltage to the minimum envelope voltage along the line:

$SWR = V_{Max}/V_{Min} = (|V_I| + |V_R|) / (|V_I| - |V_R|) = (1 + |S_{ii}|) / (1 - |S_{ii}|)$

Application: Reflection measurements with conversion of the complex S-parameter to a real SWR.

Remote command:
`CALCulate<Chn>:FORMat SWR`

Unwr Phase
Selects a Cartesian diagram with an arbitrarily scaled linear vertical axis to display the phase of the measured quantity.

Properties:

- The stimulus variable appears on the horizontal axis, scaled linearly. The phase of the complex quantity C, i.e. φ (C) = arctan (Im(C) / Re(C)), appears on the vertical axis. φ (C) is measured relative to the phase at the start of the sweep (reference phase = 0°).
- In contrast to the normal Phase format, the display range is not limited to values between –180° and +180°. This format avoids artificial jumps of the trace but can entail a relatively wide phase range if the sweep span is large.
- If the first few values of the Phase trace are close to ±180°, the "raw" unwrapped phase trace possibly jumps by almost 360° with every sweep. In order to stabilize it, the firmware selects the upper point (near +180°) as starting point, if the first few (wrapped) phase values are smaller than -178° or larger than +178°.

Application: Phase measurements, e.g. phase distortion, deviation from linearity.

Tip: After changing to the "Unwr Phase" format, use the "Auto Scale Trace" function to rescale the vertical axis and view the entire trace (see "Auto Scale Trace" on page 445).

Remote command:

`CALCulate<Chn>:FORMat UPHase`

Lin Mag

Selects a Cartesian diagram with a linear vertical axis scale to display the magnitude of the measured quantity.

Properties: The stimulus variable appears on the horizontal axis, scaled linearly. The magnitude of the complex quantity C, i.e. |C| = sqrt ($Re(C)^2$ + $Im(C)^2$), appears on the vertical axis, also scaled linearly.

Application: Real measurement data (i.e. the stability factors and the DC voltages) are always displayed in a Lin Mag diagram.

Tip (alternative formats): The magnitude of each complex quantity can be displayed on a logarithmic scale. It is possible to view the real and imaginary parts instead of the magnitude and phase.

Remote command:

`CALCulate<Chn>:FORMat MLINear`

Log Mag

Selects a Cartesian diagram with a logarithmic (base 10) vertical axis scale to display the magnitude of the measured quantity.

Properties: The stimulus variable appears on the horizontal axis, scaled linearly. The magnitude of the complex quantity C, i.e. |C| = sqrt ($Re(C)^2$ + $Im(C)^2$), appears on the vertical axis, scaled logarithmically.

Application: Impedance measurements

Remote command:

`CALCulate<Chn>:FORMat LOGarithmic`

Real

Selects a Cartesian diagram to display the real part of a complex measured quantity.

Properties: The stimulus variable appears on the horizontal axis, scaled linearly. The real part Re(C) of the complex quantity C = Re(C) + j Im(C), appears on the vertical axis, also scaled linearly.

Application: The real part of an impedance corresponds to its resistive portion.

Tip (alternative formats): It is possible to view the magnitude and phase of a complex quantity instead of the real and imaginary part. The magnitude can be displayed on a linear scale or on a logarithmic scale. Both the real and imaginary parts are displayed in the polar diagram.

Remote command:
`CALCulate<Chn>:FORMat REAL`

Imag
Selects a Cartesian diagram to display the imaginary part of a complex measured quantity.

Properties: The stimulus variable appears on the horizontal axis, scaled linearly. The imaginary part Im(C) of the complex quantity C = Re(C) + j Im(C), appears on the vertical axis, also scaled linearly.

Application: The imaginary part of an impedance corresponds to its reactive portion. Positive (negative) values represent inductive (capacitive) reactance.

Tip (alternative formats): It is possible to view the magnitude and phase of a complex quantity instead of the real and imaginary part. The magnitude can be displayed on a linear scale or on a logarithmic scale. Both the real and imaginary parts are displayed in the polar diagram.

Remote command:
`CALCulate<Chn>:FORMat IMAGinary`

Inv Smith
Selects an inverted Smith chart to display a complex quantity, primarily a reflection S-parameter.

Properties: The inverted Smith chart is a circular diagram obtained by mapping the positive complex semi-plane into a unit circle. If the measured quantity is a complex reflection coefficient S_{ii}, then the unit inverted Smith chart represents the normalized admittance. In contrast to the polar diagram, the scaling of the diagram is not linear.

Application: Reflection measurements, see example in "Inv Smith" on page 113.

Tip: The axis for the sweep variable is lost in Smith charts but the marker functions easily provide the stimulus value of any measurement point. dB values for the magnitude and other conversions can be obtained by the "Marker Format" functions.

Remote command:
`CALCulate<Chn>:FORMat ISMith`

Delay
Calculates the (group) delay from the measured quantity (typically a transmission S-parameter) and displays it in a Cartesian diagram.

Properties: The group delay τ_g represents the propagation time of wave through a device. τ_g is a real quantity and is calculated as the negative of the derivative of its phase response. A non-dispersive DUT shows a linear phase response, which produces a constant delay (a constant ratio of phase difference to frequency difference).

For more information, refer to Section 4.3.8, "Group delay", on page 134.

Application: Transmission measurements, especially with the purpose of investigating deviations from linear phase response and phase distortions. To obtain the delay, a frequency sweep must be active.

Tip: The cables between the analyzer test ports and the DUT introduce an unwanted delay, which often can be assumed to be constant. Use the Zero Delay at Marker function, define a numeric length "Offset" or use the "Auto Length" function to compensate for this effect in the measurement results. To compensate for a frequency-dependent delay in the test setup, a system error correction is required.

Note: The delay for reflection factors corresponds to the transmission time in forward and reverse direction; see "Length and delay measurement, related settings" on page 199.

Remote command:
`CALCulate<Chn>:FORMat GDELay`

Delay Derivation
Derivative of the delay, i.e. the (negative) second derivative of the phase.

Remote command:
`CALCulate<Chn>:FORMat GDDerivation`

Aperture Points
Defines an aperture Δf for the (group) "Delay" and "Delay Derivation" calculation. The value is entered as number of sweep steps (= number of sweep points - 1).

For background information, see Section 4.3.8, "Group delay", on page 134.

Properties: The delay at each sweep point is computed as:

$$\tau_{g,meas} = -\frac{\Delta\phi_{deg}}{360° \cdot \Delta f}$$

where the aperture Δf is a finite frequency interval around the sweep point f_0 and the analyzer measures the corresponding phase change ΔΦ, which is calculated via a linear regression algorithm over the frequency points of the aperture.

Application: The aperture must be adjusted to the conditions of the measurement. A small aperture increases the noise in the group delay; a large aperture tends to minimize the effects of noise and phase uncertainty, but at the expense of frequency resolution. Phase distortions (i.e. deviations from linear phase) which are narrower in frequency than the aperture tend to be smeared over and cannot be measured.

Remote command:
`CALCulate<Chn>:GDAPerture:SCOunt`

Dflt Marker Frmt
Defines the default marker format of the active trace. "Default" means formatted according to the selected trace format.

New markers are formatted with the trace's "Dflt Marker Frmt"; existing markers are reformatted if (and only if) their Marker Format is set to (Trace) "Default".

For background information on marker formats, see "Marker format" on page 99.

Remote command:
`CALCulate<Chn>:MARKer:DEFault:FORMat`

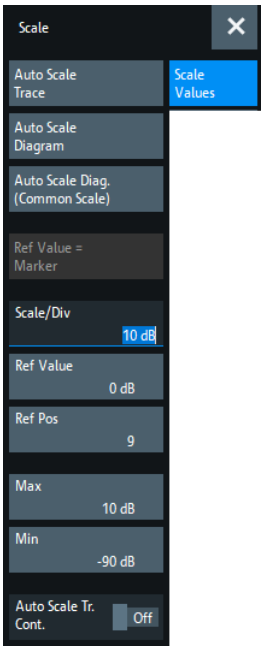
5.4 Scale softtool

The "Scale" softtool allows you to define how the active trace is displayed in its current format.

Access: Trace – [Scale] hardkey

5.4.1 Scale Values tab

Provides the functions for diagram scaling.



The "Scale Values" settings are closely related to the "Format" and "Display" settings.

The "Scale Values" settings depend on the current trace format (diagram type) because not all diagrams can be scaled in the same way:

- In Cartesian diagrams, all scale settings are available.
- In circular diagrams, no "Scale/Div", no "Ref Pos", and no "Max" and "Min" values can be defined.

The default scale is activated automatically when a display format (diagram type) is selected. Scale settings that are not compatible with the current display format are unavailable (grayed out).

Relations between the scaling parameters

The scaling parameters "Scale/Div""Ref Value", "Ref Pos", "Max", and "Min" are coupled together in the following manner:

- "Max" – "Min" = "Scale/Div" · <Number of graticule divisions>
- "Max" = "Ref Value" when "Ref Value" is 10
- "Min" = "Ref Value" when "Ref Value" is 0



Alternatives to Scaling

There are several alternatives to manual trace/diagram scaling. Refer to the following sections:

- Section 3.3.7, "Scaling diagrams", on page 65
- Section 5.8.1, "Stimulus tab", on page 526

Auto Scale Trace

Adjusts the "Scale/Div" and the "Ref Value" to display the entire active trace in the diagram area, leaving an appropriate display margin.

- In Cartesian diagrams, the analyzer recalculates the values of the vertical divisions so that the trace fits onto approx. 80% of the vertical grid. The reference value is chosen to center the trace in the diagram.
- In circular diagrams ("Polar", "Smith", "Inv Smith"), the analyzer recalculates the values of the radial divisions so that the diagram is confined to approx. 80% of the outer circumference. The reference value is set to the value of the outer circumference.

Auto scale does not affect the stimulus values and the horizontal axis.

Remote command:

```
DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:Y[:SCALe]:AUTO
```

Auto Scale Diagram

Adjusts the "Scale/Div" and the "Ref Value" to display all traces in the diagram area, leaving an appropriate display margin. All traces in the active diagram are scaled independently (see "Auto Scale Trace" on page 445), and irrespective of their channel assignment.

Remote command:

```
DISPlay[:WINDow<Wnd>]:Y[:SCALe]:AUTO
```

Auto Scale Diag. (Common Scale)

Similar to "Auto Scale Diagram" on page 445, but scales equally formatted traces together.

Remote command:

```
DISPlay[:WINDow<Wnd>]:Y[:SCALe]:AUTO COMMon
```

Ref Value = Marker

See "Ref Val = Marker / Max = Marker / Min = Marker" on page 523.

Scale/Div

Sets the value of the vertical diagram divisions in Cartesian diagrams.

"Scale/Div" corresponds to the increment between two consecutive grid lines. The unit depends on the display format: dB for display format "dB Mag", degrees for "Phase" and "Unwr Phase", ns for "Delay", U (units) for all other (dimensionless) formats.

"Scale/Div" is not available for circular diagrams ("Polar", "Smith", "Inv Smith").

Remote command:
`DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:Y[:SCALe]:PDIVision`

Ref Value
Sets the reference line of a Cartesian diagram or the outer circumference of a circular diagram.

- In Cartesian diagrams "Ref Value" defines the value of the reference line, indicated by an arrowhead symbol at the right edge of the diagram area. The color of the symbol corresponds to the trace color. As the "Ref Value" is varied, the position of the reference line ("Ref Pos") is left unchanged, so that the current trace is shifted in vertical direction. The unit of the "Ref Value" depends on the display format: dB for display format "dB Mag", degrees for "Phase" and "Unwr Phase", ns for "Delay", U (units) for all other (dimensionless) formats.
- In circular diagrams ("Polar", "Smith", "Inv Smith"), "Ref Value" defines the value of the outer circumference. Changing "Ref Value" enlarges or scales down the diagram, leaving the center unchanged. The unit is U (units) for all circular diagrams.

Remote command:
`DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:Y[:SCALe]:RLEVel`

Ref Pos
Defines the position of the reference line in a Cartesian diagram.

The reference line is indicated by an arrowhead symbol at the right edge of the diagram area. The color of the symbol corresponds to the trace color. "Ref Pos"is defined on a linear scale between 0 (bottom line of the diagram) and 10 (top line of the diagram). As the "Ref Pos" is varied, the value of the reference line ("Ref Value") is left unchanged, so the current trace is shifted together with the "Ref Pos".

"Ref Pos" is not available (grayed) for polar diagrams ("Polar", "Smith", "Inv Smith").

Remote command:
`DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:Y[:SCALe]:RPOSition`

Max / Min
Define the upper and lower edge of a Cartesian diagram.

"Max" and "Min" are not available (grayed) for polar diagrams ("Polar", "Smith", "Inv Smith").

Remote command:
`DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:Y[:SCALe]:TOP`
`DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:Y[:SCALe]:BOTTom`

Auto Scale Tr. Cont.
Similar to Auto Scale Trace, but applies auto-scaling continuously, even during a running sweep.

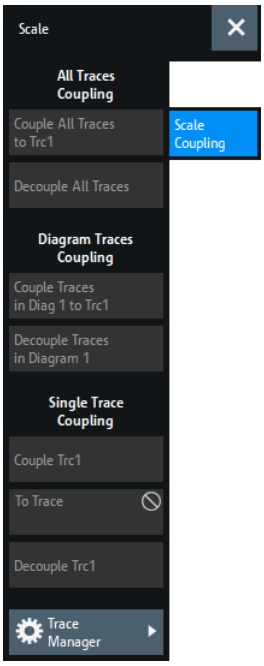
Remote command:
`DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:Y[:SCALe]:CONTinuous`

5.4.2 Scale Coupling tab

Selects common scale settings for all traces. The softkeys are available if the active recall set contains at least two traces, and if the active trace is not a reference trace ("To Trace").

Related settings

Refer to Section 5.5.1.3, "Trace Manager dialog", on page 453.



The "Trace Manager..." button opens the Trace Manager dialog.

Couple All Traces / Couple Trc ... To Trace
Applies the scale settings of the reference trace ("To Trace") to all traces / to the active trace.

Remote command:

```
DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:COUPle:ALL ON
DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:COUPle ON, <TraceName>
```

Decouple All Traces / Decouple Trc
Assigns independent scale settings to all traces / to the active trace.

Remote command:

```
DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:COUPle:ALL OFF
DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:COUPle OFF
```

5.4.3 Zoom tab

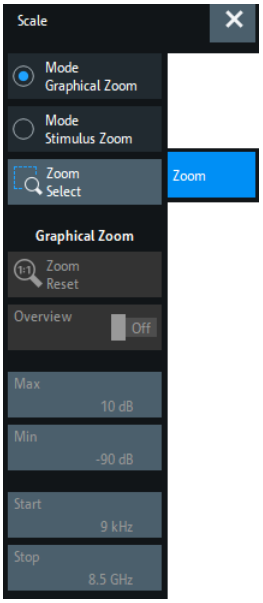
Provides graphical and stimulus zoom functions for cartesian diagrams.

The graphical zoom magnifies a (paraxial) rectangular portion of a diagram without modifying any sweep parameters. The stimulus zoom also magnifies the selected rectangle, but at the same time narrows the sweep range of the active channel to the rectangle's horizontal range.

Alternatives to Zooming

There are several alternatives to graphical/numerical zooming. Refer to the following sections:

- Section 3.3.7, "Scaling diagrams", on page 65
- Section 5.4.1, "Scale Values tab", on page 444
- Section 5.8.1, "Stimulus tab", on page 526



Mode Graphical Zoom/Mode Stimulus Zoom
Switches between graphical and stimulus zoom mode.

Use the zoom selection button to initiate the selection of a (paraxial) rectangular zoom area. Its label changes according to the selected zoom mode.

Remote command:
n.a.

Zoom Select/Stim. Zoom Select
Prepares the analyzer GUI for the selection of a (paraxial) rectangular zoom area.

The effect of the subsequent selection depends on the current zoom mode (as indicated on the button label):

- In "Mode Graphical Zoom" (button label "Zoom Select"), the selected rectangle is magnified without modifying any sweep parameters.
- In "Mode Stimulus Zoom" (button label "Stim. Zoom Select") the selected rectangle is magnified as well, but at the same time the sweep range of the active channel is narrowed to the rectangle's horizontal range.

You can define the zoom area using touchscreen or mouse. To modify the zoom window in graphical zoom mode, you can also use the numerical input fields "Max", "Min", "Start", and "Stop".

Remote command:
DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:ZOOM[:STATe]

Zoom Reset
If a graphical zoom has been applied to the current diagram, this action resets the zoom area.

Remote command:
DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:ZOOM[:STATe]

Overview On
If a graphical zoom has been applied to the active diagram (and has not been reset, this button toggles the overview for this diagram.

The overview appears in the upper part of the diagram and shows the original diagram and the zoom area. You can move the zoomed part of the trace by moving the zoom area or use the numerical input fields "Max", "Min", "Start", "Stop" to do so.

Remote command:
DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:ZOOM[:STATe]

Max / Min / Start / Stop
Defines the coordinates of the graphical zoom window for the active diagram. "Max" and "Min" define the response axis range, "Start" and "Stop" define the stimulus axis range.

The input fields are only enabled if a zoom area was selected before.

Remote command:
DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:ZOOM:BOTTom
DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:ZOOM:STARt
DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:ZOOM:STOP
DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:ZOOM:TOP

5.5 Trace Config softtool

The "Trace Config" softtool provides functions for managing traces.

Access: Trace – [Trace Config]

5.5.1 Traces tab

Provides functions to handle traces and diagram areas, and assign traces to channels.

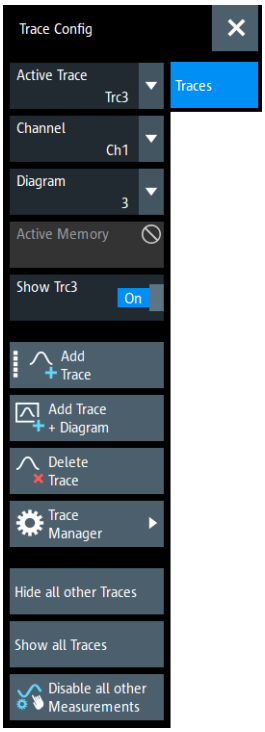
Related information

Refer to the following sections:

- Section 4.1.3, "Traces, channels and diagrams", on page 80
- Section 3.3.5, "Handling diagrams, traces, and markers", on page 57

In remote control, each channel can contain an active trace. The active remote traces and the active manual trace are independent of each other; see Section 6.3.2, "Active traces in remote control", on page 883.

5.5.1.1 Controls on the Traces tab



Active Trace

Selects an arbitrary trace of the active recall set as the active trace in its channel and diagram. At the same time, it sets the trace's diagram and channel as the active Channel and Diagram.

Tip: You can also select an item in a trace list or a trace line in a diagram to make the related trace the active one.

This function is disabled if only one trace is defined.

Remote command:

`CALCulate<Ch>:PARameter:SELect`

Channel

Allows you to move the Active Trace to a different channel.

Remote command:
CALCulate:PARameter:MOVE

Diagram
Allows you to move the Active Trace to another diagram.

Remote command:
DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:FEED

Active Memory
Selects one of the memory traces of (or related to) the active trace as its (or its parent data trace's) active memory trace.

Remote command:
CONFigure:TRACe:MEM:ACTive

Show
Toggles the visibility of the Active Trace

Remote command:
DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:SHOW

Add Trace
Copies the active trace (within the active channel), assigns it to the same diagram, and makes the new trace the active one.

"Copies" means that the new trace is created with the settings of the former active trace, i.e. the former and the new active trace overlay each other. Change the reference position or select a different measurement for the new trace to separate them (see Section 5.2, "Meas softtool", on page 291).

The new trace is named "Trc<n>", where <n> is the smallest positive integer such that this name is not already used. The name can be changed in the Trace Manager dialog.

If you drag and drop the "Add Trace" button to a diagram area, the firmware opens the New Trace dialog that lets you select a different measurement.

Remote command:
CALCulate<Ch>:PARameter:COPY Trc<n>[, 0]

Add Tr+Diag
Copies the active trace (within the active channel), assigns it to a new diagram, and makes the new trace the active one.

"Copies" means that the new trace is created with the settings of the former active trace. It is named "Trc<n>", where <n> is the smallest positive integer such that this name is not already used. The name can be changed in the Trace Manager dialog.

Remote command:
CALCulate<Ch>:PARameter:COPY Trc<n>, 1

Delete Trace
Deletes the active trace and removes it from the diagram area. If the active diagram contains only one trace, the diagram is also deleted.

"Delete Trace" is disabled if the active recall set contains only one trace. In manual control, each recall set must contain at least one diagram area with one channel and one trace.

Tips:

- You can also hide traces without actually deleting them: remove the corresponding "On" flags in the "Trace Manager" (see Section 5.5.1.3, "Trace Manager dialog", on page 453).
- Use the undo function to restore a trace that was unintentionally deleted.

Remote command:

```
CALCulate<Ch>:PARameter:DELete
CALCulate:PARameter:DELete:ALL
CALCulate<Ch>:PARameter:DELete:CALL
```

Trace Manager

Opens the Trace Manager dialog.

Hide All Other Traces / Show All Traces

Configures the visibility of the traces in the active diagram:

- "Hide All Other Traces" hides all traces of the active diagram – except the active one (which can be visible or not)
- "Show All Traces" makes all traces of the active diagram visible

Note:

- These actions can also be performed from the context menu of the trace name segment in the trace list (see Section 4.2.1.4, "Channel list and channel settings", on page 103).
- Use the "On" flags in the "Trace Manager" to show/hide arbitrary sets of traces (see Section 5.5.1.3, "Trace Manager dialog", on page 453).

Disable all other Measurements/Enable all Measurements

Toggle button with alternating caption.

Same functionality as the / toggle button of the Section 3.3.2.2, "Toolbar", on page 49

"Disable all other Measurements"	• Disables all channels, except the active one • Disables all traces in the active channel, except the active one • Hides all diagrams except the active one
"Enable all Measurements"	Restores the previous channel/trace/diagram configuration

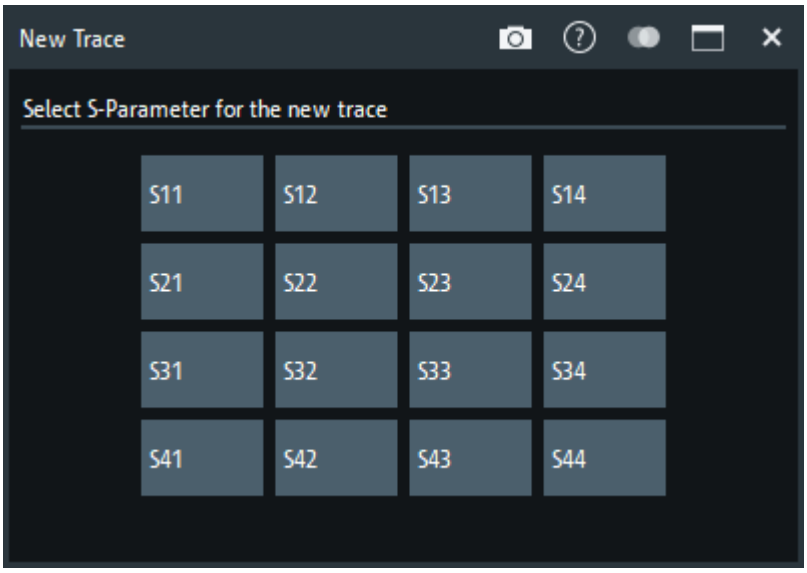
Same functionality as the

5.5.1.2 New Trace dialog

The tool bar button allows you to create a trace in the active channel.

- Tap/click the button to duplicate the active trace (equivalent to Add Trace).
- Drag the respective button onto the diagram area to create the trace in any other existing diagram or in a new one.

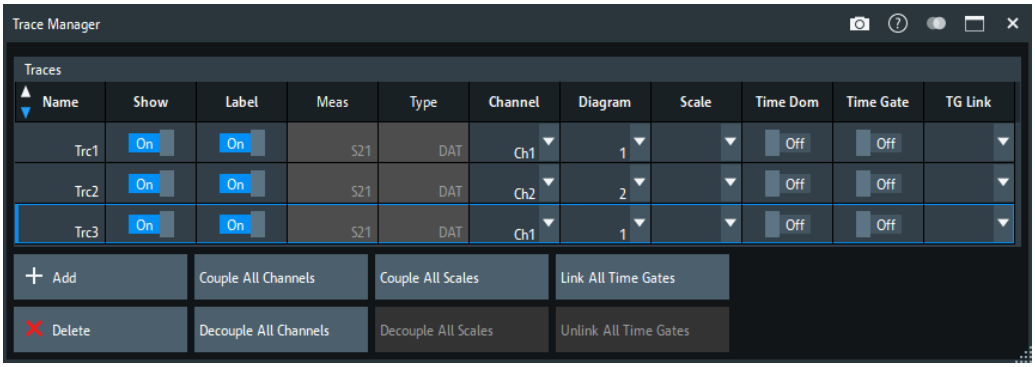
After the button has been dropped, the "New Trace" dialog pops up and lets you select the quantity to be measured.



5.5.1.3 Trace Manager dialog

The "Trace Manager" dialog allows you to perform operations on traces.

Access: Trace – [Trace Config] > "Traces" > "Trace Manager"



All existing traces of the current recall set are listed in a table with several editable (white) or non-editable (gray) columns.

Table Area

The table contains the following columns:

- "Name" indicates name of the related trace.
  Trace names must be unambiguous across all channels and diagram areas in a recall set.
- "On" indicates and controls the visibility of the related trace.
- "Label" indicates and controls the visibility of the related trace label
- "Meas" indicates the measured parameter.

- "Type" indicates whether the trace is a data trace ("DAT"), displaying the current measurement data, or a memory trace ("MEM").
- "Channel" indicates and controls the channel to which the related trace is assigned. Data traces and their associated memory traces are always assigned to the same channel.
- "Diagram" indicates and controls the diagram area to which the related trace is assigned.
- "Scale" indicates and controls the scale coupling of the related trace.
  A trace's scaling can either be uncoupled ("Scale" empty) or coupled to another trace's scaling.
- "Time Dom", "Time Gate" and "TG Link" are only available if the Time domain analysis option is installed.
  - "Time Dom" enables time domain representation of the related trace. See "Time Domain" on page 357
  - "Time Gate" enables time gating. See "Time Gate" on page 364
  - "TG Link" configures time gate linking.
    See "Time Gate Linking" on page 365.

Rules for trace names

The analyzer can define mathematical relations between different traces and calculate new mathematical traces ("User Def Math"). The trace names are used as operands in the mathematical expressions and must be distinguished from the mathematical operators +, -, *, /, (,) etc., which places some restrictions on the syntax of trace names.

- The first character of a trace name can be one of the following:
  - an upper case letter from A to Z, or lower case letter from a to z
  - an underscore _
  - a square bracket [or]
- For all other characters of a trace name, the numbers 0 to 9 can be used in addition.

Note: The analyzer does not accept illegal or ambiguous trace names. If an illegal or ambiguous name is specified, the entry is denied.

Remote command:

```
CONFigure:TRACe:CATalog?
CONFigure:CHANnel<Ch>:TRACe:CATalog?
CONFigure:TRACe<Trc>:REName
CONFigure:CHANnel<Ch>:TRACe:REName
CONFigure:CHANnel<Ch>:NAME:ID?
CALCulate:PARameter:MOVE
DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:SHOW
DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:LABel:SHOW
CONFigure:TRACe:WINDow:TRACe?
DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:FEED
```

Add

Creates a trace based on the Active Trace. In particular, the trace is assigned to the channel and diagram of the active trace. However, its "Scale" coupling is not adopted.

The default names for new traces are "Trc<n>", where <n> is selected by the analyzer firmware to make trace names unambiguous.

Remote command:
CALCulate<Ch>:PARameter:SDEFine

Delete
Deletes the selected trace.

This button is disabled if the recall set contains only one trace: In manual control, each recall set must contain at least one diagram area with one channel and one trace.

Remote command:
CALCulate<Ch>:PARameter:DELete

Couple All Channels / Decouple All Channels
- "Couple All Channels" assigns all traces to the channel of the active trace, deleting all other (now unused) channels. The analyzer displays a confirmation dialog box before deleting the unused channels.
- "Decouple All Channels" makes sure that each data trace is assigned its own (independent) channel.
  For data traces previously assigned to the same channel, new channels are created based on the original channel's settings. Data traces and their associated memory traces are assigned to the same channel.

Remote command:
n/a

Couple All Scales / Decouple All Scales
- "Couple All Scales" couples the scale settings of all traces to the scale settings of the active trace. The scale settings of the other traces are lost.
- "Decouple All Scales" applies independent scale settings to all traces.
  If trace A is coupled to trace B, then B's scale settings are copied to A.

Remote command:
n/a

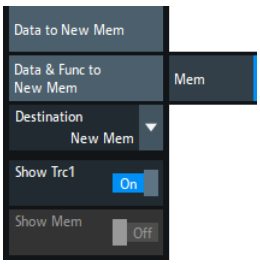
Link All Time Gates/Unlink All Time Gates
Only available if the Time domain analysis option is installed.

See "Time Gate Linking" on page 365.

5.5.2 Mem tab

Stores traces to the memory.

Background information
Refer to "Trace types" on page 95.



Coupling of data and memory traces

When a memory trace is generated from a data trace, it is displayed in the same diagram area and inherits all channel and trace settings from the data trace. The memory trace displayed in the active diagram; its properties are indicated in the trace list:



New memory traces are named "Mem<n>[<Data Trace>]", where:

- <n> counts all data and memory traces in the active recall set in chronological order
- <Data_Trace> is the name of the associated data trace

Trace names can be changed in the Trace Manager dialog.

The following display settings of a data trace and the associated memory traces are **fully coupled**. Changing a property of one trace affects the properties of all other traces.

- All "Format" settings (see Section 5.3, "Format softtool", on page 438)
- All "Scale" settings (see Section 5.4, "Scale softtool", on page 444)

Selection of the measured quantity (using the Meas softtool) is possible for the data trace but disabled for the memory traces.

Channel settings made for a memory trace act on the associated data trace. Some of the channel settings for a data trace (e.g. the "Stimulus" range) also affect the display of the memory traces.

If, due to a change of the sweep type, the stimulus type of a data trace changes, all its memory traces are deleted.

Active Trace vs. Active Data Trace

In the context of memory traces we distinguish between the active trace and the active data trace.

- If the active trace is a memory trace, then the active data trace is the data trace to which the memory trace is associated.
- If the active trace is a data trace, then the active trace is also the active data trace.

Data to <Destination>
Stores the current state of the active data trace to the Destination memory trace. No trace functions are applied to the stored trace.

Tips:
- Use Data & Func to <Destination> to apply trace functions to the stored trace.
- You can also create memory traces using the Import Complex Data dialog.
- It is not possible to store Hold traces to memory.
- For the relation between a data trace and its associated memory traces, see "Coupling of data and memory traces" on page 456.

Remote command:
```
CALCulate<Chn>:MATH:MEMorize
TRACe:COPY
```

Data & Func to <Destination>
Stores the current state of the active data trace – including trace functions – to the Destination memory trace.

Trace functions

The trace functions comprise the following mathematical operations:
- Active trace math on unformatted and formatted traces, as configured in the Section 5.5.4, "Math tab", on page 459.
- A shift of the data trace (see "Shift Trace" on page 473).

Data to <Destination> stores the raw trace without the trace functions, "Data & Func to <Destination>" stores the trace after it has been transformed using the trace functions.

For the relation between a data trace and its associated memory traces, see "Coupling of data and memory traces" on page 456.

Remote command:
```
CALCulate<Chn>:MATH:MEMorize
TRACe:COPY:MATH
```

Destination
Selects the destination for the Data to <Destination> and Data & Func to <Destination> operations.

The destination can be one of the following:
- An existing memory trace of the active data trace.
  The existing memory trace is overwritten.
- "New Trace"
  The data are copied to a new memory trace, associated to the current data trace.

Remote command:
n/a

Show <Active Data Trace>
Shows or hides the active data trace in the diagram.

Remote command:
```
DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:SHOW
```

Show <Mem>
Shows or hides the active memory trace or the first memory trace of the active data trace.

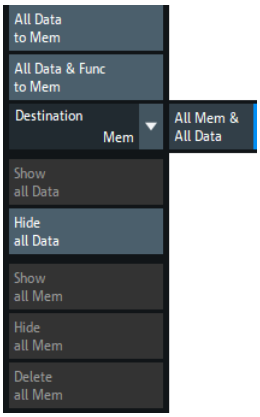
If no memory trace is associated with the active data trace, "Show <Mem>" is disabled.

Remote command:
DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:SHOW

5.5.3 All Mem & All Data tab

Performs actions on all data or memory traces in the active recall set.

Background information
Refer to "Trace types" on page 95.



All Data to <Destination>
Stores the current data of all data traces in the active recall set to memory traces, in accordance with the Destination setting. No trace functions are applied to the stored traces.

Remote command:
TRACe:COPY

All Data & Func to <Destination>
Stores the current data of all data traces in the active recall set to memory traces, in accordance with the Destination setting. Trace functions are applied to the stored traces.

For information on trace functions, see "Data & Func to <Destination>" on page 457.

All Data to <Destination> stores the raw trace without the trace functions, "All Data & Func to <Destination>" stores the trace after it has been transformed using trace functions.

Remote command:
CALCulate<Chn>:MATH:MEMorize

Destination
Selects the destination for the All Data to <Destination> and All Data & Func to <Destination> functions, that operate on all data traces in the active recall set.

- **"Mem"**:

For each data trace with associated memory traces, the current trace data are copied to the first associated memory trace, overwriting existing data. For data traces without associated memory trace, the current trace data are copied to a new memory trace, associated to this data trace.

- **"New"**:
  For each data trace, the current trace data are copied to a new memory trace, associated to this data trace.

New memory traces are named "Mem<n>[<Data_Trace>]" with <n> selected by the analyzer firmware to make trace names unique.

Remote command:
n/a

Show All Data / Hide All Data / Show All Mem / Hide All Mem
Displays or hides all data or memory traces in the active recall set. Hidden traces are not deleted.

Remote command:
`DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:SHOW`

Delete All Mem
Deletes all memory traces in the active recall set.

Tips:
- Use the Trace Manager dialog to hide or delete arbitrary sets of traces.
- Use the UNDO function of the analyzer to restore a trace that was unintentionally deleted.

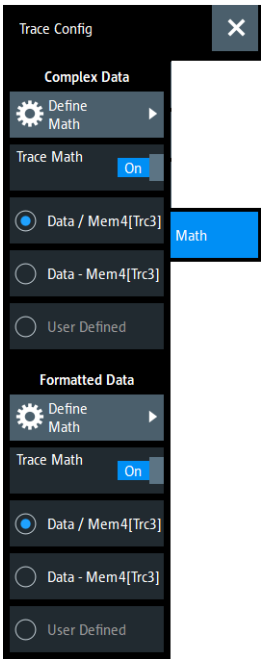
Remote command:
`CALCulate:PARameter:DELete:MEMory`

5.5.4 Math tab

Allows you to define mathematical operations on traces.

Background information
Refer to "Trace types" on page 95.

5.5.4.1 Controls on the Math tab



The "Define Math" buttons in the "Complex Data" and "Formatted Data" sections both open the User Def Math dialog, but with different scope:

- "Complex Data" > "Define Math" defines mathematical operations on raw complex trace data.
- "Formatted Data" > "Define Math" defines mathematical operations on formatted trace data.

Both can be combined.

Complex Data/Formatted Data
The controls in the "Complex Data" section define math operations on raw complex trace data, the controls in the "Formatted Data" section define math operations on formatted trace data.

"Complex Data" math is available in all firmware versions of all R&S ZNx vector network analyzers. "Formatted Data" math was added in version 3.0 of the R&S ZNB firmware.

Trace Math ← Complex Data/Formatted Data
Activates or deactivates the "Complex Data" and/or "Formatted Data" formula previously defined using Data / <Mem>, Data - <Mem>, or "Define Math" (via Section 5.5.4.2, "User Def Math dialog", on page 461).

Trace math for unformatted and formatted trace data can be active at the same time.

Trace Math ← Complex Data/Formatted Data
Activates the mathematical mode, applying the last active mathematical relation to the active trace. The trace is replaced by the mathematical trace and Math is displayed in the trace list while the mathematical mode is active:



"Trace Math" is enabled if the active data trace fulfills the conditions for evaluating the mathematical relation. E.g., if no "User Defined" mathematical relation is defined, a memory trace must be coupled to the active data trace, so that the R&S ZNB can evaluate one of the relations "Data / <Mem>" or "Data - <Mem>".

Remote command:
`CALCulate<Chn>:MATH:STATe`
`CALCulate<Chn>:MATH:FORMatted:STATe`

Data / <Mem>, Data - <Mem> ← Complex Data/Formatted Data
Activates the mathematical mode with the corresponding trace mathematical operation. The division (subtraction) is calculated on a point-to-point basis: Each measurement point of the active trace is divided by (subtracted from) the corresponding measurement point of the memory trace. If the memory trace represents the result of a previous sweep with unchanged settings, the divided (subtracted) curve is typically centered at 1 / 0 dB (0). It shows the variation of the results in subsequent sweeps.

The result of the division is a mathematical trace and replaces the active data trace in the diagram area. The mathematical trace is updated as the measurement goes on and the analyzer provides new active trace data.

This function is disabled unless a memory trace is coupled to the active data trace. Trace coupling ensures that the two traces have the same number of points so that the mathematical trace is well-defined.

Remote command:
`CALCulate<Chn>:MATH:FUNCtion`
`CALCulate<Chn>:MATH:FORMatted:FUNCtion`

User Defined ← Complex Data/Formatted Data
Activates the mathematical mode and displays the mathematical trace defined using in the "User Def Math" dialog (see Section 5.5.4.2, "User Def Math dialog", on page 461).

The mathematical trace replaces the active data trace in the diagram area; it is updated as the measurement goes on and the analyzer provides new active trace data.

Remote command:
`CALCulate<Chn>:MATH:STATe`

5.5.4.2 User Def Math dialog

The "User Def Math" dialog defines a mathematical trace for the active trace. Each measurement point of the original trace is replaced by the corresponding point of the mathematical trace.

Access: Trace – [Trace Config] > Math > Complex Data/Formatted Data section > "Define Math"

Depending on the section from where the dialog is opened, the calculation is either based on unformatted or formatted trace data.

Compatibility between traces in mathematical relations

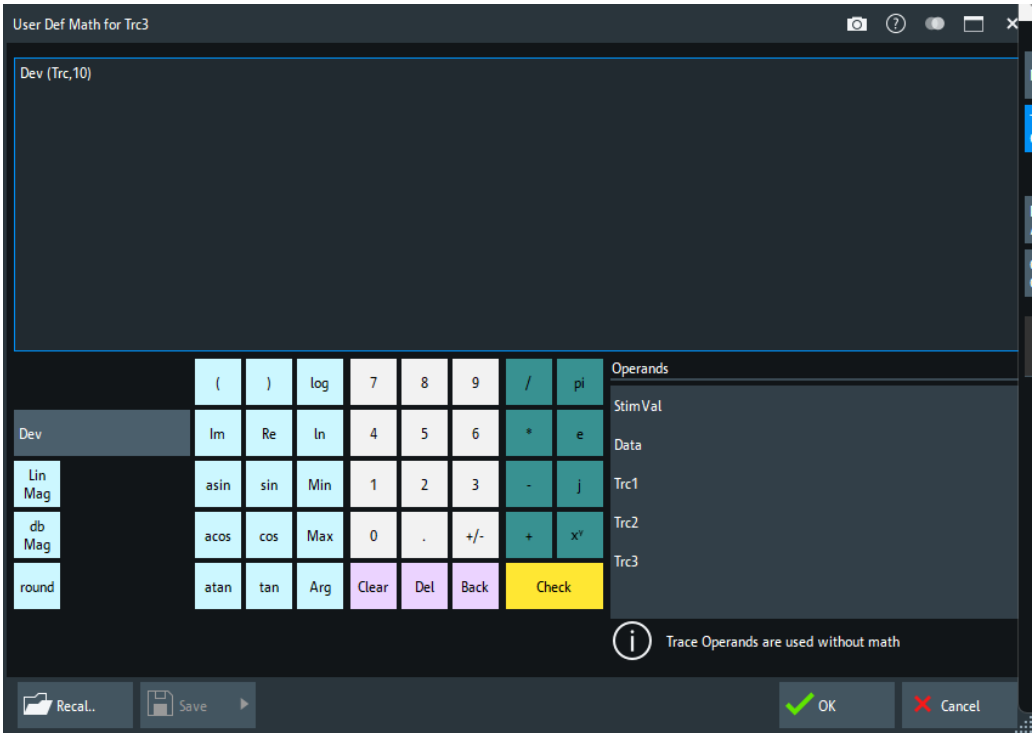
Mathematical traces are either constant functions or functions of one or more data or memory traces. They are calculated on a point-to-point basis. Each trace point no. i of the mathematical trace is calculated from a set of constant values c_1, ..., c_n plus the trace points $Trc1_i$, $Trcm_i$ of all traces 1 to m in the mathematical relation:

$Math_i = Fct. (c_1, ..., c_n, Trc1_i, Trcm_i), i = 1, no. of points$

Different traces can be used in the same mathematical relation if they contain the same number of points. The analyzer places no further restriction on the compatibility of traces, e.g. the sweep points of the traces do not have to be the same.

The number of points belongs to the channel settings. Coupled data and memory traces are always compatible because they have the same channel settings.

The analyzer processes only numeric values without units in the mathematical formulas. No consistency check for units is performed.



Expression builder

The mathematical expression appears in the upper part of the dialog. The operands and operators in the expression can be selected from a keyboard and the list of "Operands":

- The keyboard supports the entry of numeric values, constants, and mathematical functions. In addition to the numbers 0 to 9, the decimal point and the constants j (complex unit), pi (approx. 3.14159) and e (approx. 2.71828), it contains the following buttons:
  – +/- changes the sign

  - The effect of the basic arithmetic operators (/, *, –, +) and the mathematical functions is described in Table 5-5.
    Products of numbers and constants can be entered in abbreviated form, e.g. 2e for 2*e.
  - The Clear, Del, Back buttons are used to correct faulty entries.
  - Check performs a consistency check of the displayed mathematical expression and displays a message.
- "Operands" contains all data traces and memory traces of the active recall set.
  - Data and memory traces are identified by their trace names.
  - "Data" denotes the active data trace.
  - "Mem" is the memory trace associated with the active data trace (or the first created one, if several memory traces are associated with the active data trace).
  - "StimVal" is the array of stimulus values; see footnote for Table 5-5.

  The trace operands denote *unmodified* data and memory traces. Trace math and other trace functions ("Smoothing", "Hold", "Shift Trace" etc.) are not taken into account.

Table 5-5: Effect of the operators on a complex quantity z = x + jy.

+, -, *, /	Basic arithmetic operations
()	Grouping parts of an expression
Dev(Trc, aperture)	Derivative (formatted, cartesian traces only) The derivative at stimulus *s* is calculated as the slope of the regression line between the trace points at *s* and its *aperture* neighboring stimuli (⌈1/2 *aperture*⌉ below, and ⌊1/2 *aperture*⌋ above, if possible)
Lin Mag	|z| = sqrt ($x^2 + y^2$)
dB Mag	dB Mag(z) = 20 * log|z| dB
round	x real ⇒ round(x) rounds x to the closest integer (away from zero for ties) x,y real ⇒ round(x+yj) = round(x)+round(y)j
x^y	Exponential, e.g. z^2
Arg	Phase φ (z) = arctan (Im(z) / Re(z))
Re, Im	x, y (Real and Imag)
log, ln	Common (base 10) or natural (base e) logarithm
Min, Max	Smaller or larger values of all points of two traces, e.g. Min(Trc1,Trc2)
StimVal *)	Stimulus value*)
tan, atan, sin, asin, cos, acos	Direct and inverse trigonometric functions.

*) The operand "StimVal" can be used for all sweep types. Please note that – as with all user math operands – only the numerical value without unit is processed in the user math formula.

- In frequency sweeps "StimVal" provides the stimulus frequency in Hz.
- In power sweeps, "StimVal" provides the voltage in V that results from the source power in dBm. To obtain the correct source power in dBm (for "dB Mag" trace for-

mat), Result is Wave Quantity must be enabled. Note that, due to the conversion into a dBm value, the source power depends on the reference impedance of the port associated with the measured wave quantity, to be set in the "Balanced Ports" dialog.
- In time sweeps, "StimVal" is the stimulus time in s.
- In CW mode sweeps, "StimVal" is the number of the point.

Remote command:
`CALCulate<Chn>:MATH[:EXPRession]:SDEFine`
`CALCulate<Chn>:MATH:FORMatted[:EXPRession]:SDEFine`

Result is Wave Quantity
"Result is Wave Quantity" controls the conversion and formatting of the mathematic expression.

This switch is available for **unformatted** trace data (Complex Data) only.

- If "Result is Wave Quantity" is enabled, the analyzer assumes that the result of the mathematical expression represents a voltage. Examples for voltage-type expressions are all terms proportional to a wave quantity (e.g. 1.1*Data, if a wave quantity is measured) or to a stimulus value of a power sweep.
  If "Show as": "Power" is selected in the "More Wave Quantities" dialog, the result is converted into a linear power before the selected trace format is applied. Otherwise no conversion is performed, and "dB Mag" results are referenced to 1 µV.
- If "Result is Wave Quantity" is disabled, the analyzer assumes that the result of the mathematical expression is dimensionless. Examples for dimensionless expressions are all terms proportional to ratios of wave quantities, e.g. Data / Mem2[Trc1]. The selected trace format is applied without previous conversion.

"Result is Wave Quantity" acts on the result of the mathematical expression only. Wave quantities and power sweep stimulus values always enter into the expression as voltages.

Effect of "Result is Wave Quantity"

In the More Wave Quantities dialog, the "Show as" control element specifies whether wave quantities are displayed as voltages or equivalent power values, using the port impedances for a conversion between the two representations. "Result is Wave Quantity" is relevant for mathematical traces displayed in units of dBm ("Show as": "Power" and trace format "dB Mag"):

If "Result is Wave Quantity" is on (enabled), the mathematical trace values <W> are interpreted as voltages and first converted into equivalent powers (<W> —> <P> = $<W>^2/Re(Z_0)$). Results in "dB Mag" format are calculated according to $<P>_{log}$ = 10 * log (<P>/1mW).
- If "Result is Wave Quantity" is off, the mathematical trace values <W> are interpreted as dimensionless quantities. Results in "dB Mag" format are calculated according to $<W>_{log}$ = 20 * log (<W>).

Example:

A mathematical trace value amounts to 1 (real value); the port impedance is 50 Ω. If "Result is Wave Quantity" is on, the analyzer assumes the trace value to be 1 V, which is converted into a linear power of 20 mW, corresponding to approx. 13 dBm. With "Result is Wave Quantity" off, the trace value 1 is directly converted into a logarithmic power of 0 dBm.

Tip: See also example for `CALCulate<Chn>:MATH:WUNit:STATe`.

Remote command:
`CALCulate<Chn>:MATH:WUNit[:STATe]`

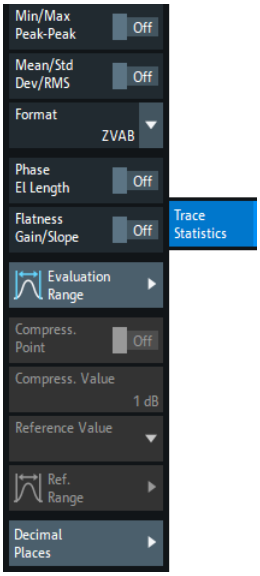
Recall / Save
Recalls / saves a mathematical expression from / to a trace math string file. Trace math string files are ASCII files with the default extension `*.mth` and contain the mathematical expression as it is written in the "User Def Math" dialog. It is possible to change or create math string files using a text editor.

Remote command:
`CALCulate<Chn>:MATH:WUNit[:STATe]`

5.5.5 Trace Statistics tab

Evaluates statistical and phase information of the entire trace or of a specific evaluation range and calculates the x-dB compression point.

5.5.5.1 Controls on the Trace Statistics tab

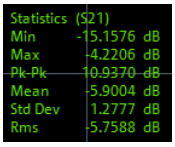


The "Evaluation Range..." button opens the "Evaluation Range" dialog (see Section 5.5.5.2, "Evaluation Range dialog", on page 470).

The "Decimal Places" button opens the "System Config" dialog to define the (maximum) number of fractional digits for setting values and measurement results. See also "User Interface tab" on page 815.

Min/Max/Peak-Peak, Mean/Std Dev/RMS

The upper two softkeys in the "Trace Statistics" tab display or hide groups of statistical results. The values are based on all response values of the trace in the selected evaluation range ("Evaluation Range...").



Suppose that the trace in the evaluation range contains n stimulus values x_i and n corresponding response values y_i (measurement points). The statistical values are obtained as follows:

- "Min" and "Max" are the largest and the smallest of all response values y_i.
- "Pk-Pk" is the peak-to-peak value and is equal to the difference "Max"–"Min"
- "Mean" is the arithmetic mean value of all response values:

$$Mean = \frac{1}{n}\sum_{i=1}^{n} y_i$$

- "Std Dev" is the standard deviation of all response values:

$$Std.Dev. = \sqrt{\frac{1}{n}\sum_{i=1}^{n}(y_i - \frac{1}{n}\sum_{i=1}^{n} y_i)^2}$$

- "RMS" is the root mean square (effective value) of all response values:

$$RMS = \sqrt{\frac{1}{n}\sum_{i=1}^{n} |y_i|^2}$$

Note: To calculate the "Min", "Max", "Pk-Pk" and the "Std Dev" values, the analyzer uses formatted response values y_i (see Section 4.2.3, "Trace formats", on page 108). To calculate the "RMS" value, the analyzer uses formatted response values for trace formats "Phase", "Unwrap Phase" and "Delay", and linear, unformatted values otherwise. In the latter case, the physical unit for unformatted wave quantities is 1 Volt, the RMS value has zero phase, and the selected trace format is applied to the unformatted RMS value.

Consequently, a trace's mean value, standard deviation and RMS value depend on the selected trace format.

Remote command:

```
CALCulate<Chn>:STATistics:MMPTpeak[:STATe]
CALCulate<Chn>:STATistics:MSTDdev[:STATe]
CALCulate<Chn>:STATistics:RMS[:STATe]
CALCulate<Chn>:STATistics:RESult?
CALCulate<Chn>:STATistics[:STATe]
CALCulate<Chn>:STATistics[:STATe]:AREA
```

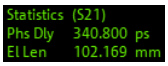
Format
This setting determines how Min/Max/Peak-Peak, Mean/Std Dev/RMS for complex-valued traces (Smith, Polar) are calculated:

- "ZVAB": the results are based on unformatted wave quantities (voltages)
- "R + jX": the results are based on the impedance values R and X
- "G + jB": the results are based on the admittance values G and B

In the two latter cases, the "RMS" value is not displayed.

Remote command:
`CALCulate<Chn>:STATistics:FORMat`

Phase/El Length
Displays or hides the phase delay ("Phs Dly") and the electrical length ("El Len") of the trace in the selected evaluation range ("Evaluation Range..."). The parameters are only available for trace formats that contain phase information, i.e. for the formats "Phase", "Unwr Phase", and the polar diagram formats "Polar", "Smith", "Inv Smith" (see Section 5.3, "Format softtool", on page 438). Moreover, the sweep type must be a frequency sweep, and the evaluation range must contain at least 3 measurement points.



The phase parameters are obtained from an approximation to the derivative of the phase in the selected evaluation range.

- "Phs Dly" is the phase delay, which is an approximation to the group delay and calculated as follows:

$$PD = -\frac{\Delta\phi_{deg}}{360° \cdot \Delta f}$$

  where Δf is the width of the evaluation range and ΔΦ is the corresponding phase change. See also note on transmission and reflection parameters below.
- "El Len" is the electrical length, which is the product of the phase delay times the speed of light in the vacuum.

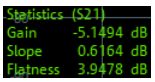
If no dispersion occurs, the phase delay is equal to the group delay. For more information, refer to Section 4.3.8, "Group delay", on page 134.

Note: To account for the propagation in both directions, delay and electrical length of a reflection parameter are only half the delay and electrical length of a transmission parameter. The formula for PD above is for transmission parameters. See the section on "Length and delay measurement" in Section 4.6.1.3, "Auto Length", on page 199.

Tip: The phase evaluation can cause misleading results if the evaluation range contains a 360 deg phase jump. The trace format "Unwr Phase" avoids this behavior.

Remote command:
`CALCulate<Chn>:STATistics:EPDelay[:STATe]`
`CALCulate<Chn>:STATistics:RESult?`
`CALCulate<Chn>:STATistics[:STATe]`

Flatness/Gain/Slope
Displays or hides trace parameters that the analyzer calculates for the selected evaluation range ("Evaluation Range...").



Suppose that A and B denote the trace points at the beginning and at the end of the evaluation range, respectively.

- "Gain" is the larger of the two stimulus values of points A and B.
- "Slope" is the difference of the stimulus values of point B minus point A.
- "Flatness" is a measure of the deviation of the trace in the evaluation range from linearity. The analyzer calculates the difference trace between the active trace and the straight line between points A and B. The flatness is the difference between the largest and the smallest response value of this difference trace.

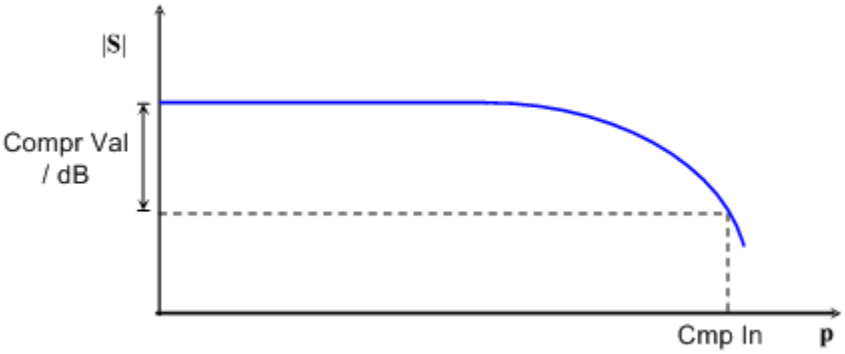
Remote command:

```
CALCulate<Chn>:STATistics:SFLatness[:STATe]
CALCulate<Chn>:STATistics:RESult?
CALCulate<Chn>:STATistics[:STATe]
CALCulate<Chn>:STATistics[:STATe]:AREA
```

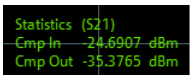
Compr. Point / Compr. Val.

Displays or hides all results related to the x dB (or x degrees) compression point of the trace, where x is the selected compression value. To obtain valid compression point results, a power sweep must be active, and the trace format must be dB or phase.

For dB traces, the x dB compression point of an S parameter or ratio is defined as the stimulus level where the response value has dropped by x dB compared to the response value at small stimulus signal levels ("small signal value"). For phase traces, the x degree compression point of an S parameter or ratio is defined as the stimulus level where the phase value has dropped by x° compared to the phase value at small stimulus signal levels. By default, the analyzer uses the value at the "First Point" of the selected evaluation range ("Evaluation Range...") as an approximation for the small signal value. However, you may also choose a different Reference Value.



The compression point is a measure for the upper edge of the linearity range of a DUT. It is close to the highest input signal level for which the DUT shows a linear response, so that the magnitude of all S-parameters remains constant).





- "Cmp In" is the stimulus level at the compression point in units of dBm. "Cmp In" always corresponds to the driving port level (e.g. the level from port no. j, if a transmission parameter S_{ij} is measured).
- For dB formatted traces, "Cmp Out" is the sum of the stimulus level "Cmp In" and the magnitude of the measured response value at the compression point. The magnitude of a transmission S-parameter S_{ij} is a measure for the attenuation (or gain) of the DUT, hence: "Cmp Out" = "Cmp In" + <Attenuation>.
  For phase formatted traces, "Cmp Out" is the response value at the compression point.

The info field shows invalid results ('----') if the wrong sweep type or trace format is selected, or if no compression point with the configured properties was found.

Remote command:
```
CALCulate<Chn>:STATistics:NLINear:COMP[:STATe]
CALCulate<Chn>:STATistics:NLINear:COMP:LEVel
CALCulate<Chn>:STATistics:NLINear:COMP:PHASe
CALCulate<Chn>:STATistics:NLINear:COMP:RESult?
CALCulate<Chn>:STATistics[:STATe]:AREA
```

Reference Value
This control is only available if compression point statistics are calculated. It selects the reference value ("small signal value") for the compression point calculation.

"First Point" (default)	uses the value at the first point of the selected evaluation range as the reference value (see Section 5.5.5.2, "Evaluation Range dialog", on page 470).
"Selected Marker"	uses the value at a selectable marker as the reference value (see "Selected Marker" on page 469).
"Range"	uses the average value in a configurable "Ref. Range" on page 469 as the reference value.
"Defined Value"	allows you to specify the reference value manually (see "Defined Value" on page 470).

Remote command:
```
CALCulate<Chn>:STATistics:NLINear:COMP:REFerence
```

Selected Marker
This button is only visible if "Selected Marker" is used as Reference Value. It allows you to select the marker whose value shall be used as the reference ("small signal value") for the compression point calculation.

Remote command:
```
CALCulate<Chn>:STATistics:NLINear:COMP:RMARker
```

Ref. Range
This button is only visible if "Range" is used as Reference Value.

It opens the "Reference Range" dialog that allows you to configure the sweep range whose average value shall be used as the reference ("small signal value") for the compression point calculation.

Remote command:
CALCulate<Chn>:STATistics:NLINear:COMP:RDOMain:USER
CALCulate<Chn>:STATistics:NLINear:COMP:RDOMain:USER:STARt
CALCulate<Chn>:STATistics:NLINear:COMP:RDOMain:USER:STOP
CALCulate<Chn>:STATistics:NLINear:COMP:RDOMain:USER:SHOW

Defined Value
This setting is only visible if "Defined Value" is used as Reference Value. It manually defines the reference value ("small signal value") for the compression point calculation.

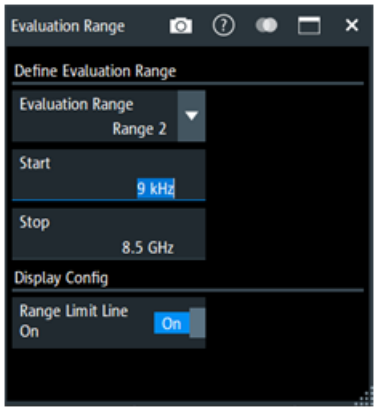
Remote command:
CALCulate<Chn>:STATistics:NLINear:COMP:RLEVel
CALCulate<Chn>:STATistics:NLINear:COMP:RPHase

5.5.5.2 Evaluation Range dialog

The "Evaluation Range" dialog defines the range for various trace evaluation operations (trace statistics, offset, loss and fixture compensation calculations). The evaluation range is a continuous interval of the sweep variable.

Access:

- Trace – [Trace Config] > "Trace Statistics" > "Evaluation Range..."
- Channel – [Offset Embed] > "Offset" > "Evaluation Range..."
- ...



Evaluation Range
Selects a predefined evaluation range. Up to 10 different ranges are available for each recall set. "Full Span" means that the search range is equal to the entire sweep range. The trace statistics functions consider all measurement points with stimulus values x_i between the "Start" and "Stop" value of the evaluation range:

"Start" ≦ x_i ≦ "Stop"

The evaluation ranges are defined similar to the marker search ranges. For more information, see Section 5.7.3.2, "Search Range dialog", on page 511.

Note: A restricted evaluation range is indicated in the "Trace Statistics" info field.



Remote command:

```
CALCulate<Chn>:STATistics:DOMain:USER
CALCulate<Chn>:STATistics:DOMain:USER:STARt
CALCulate<Chn>:STATistics:DOMain:USER:STOP
```

Range Limit Lines On

Displays or hides the range limit lines in the diagram area. Range limit lines are two vertical lines at the Start and Stop values of the active evaluation range ("Range 1" to "Range 10").

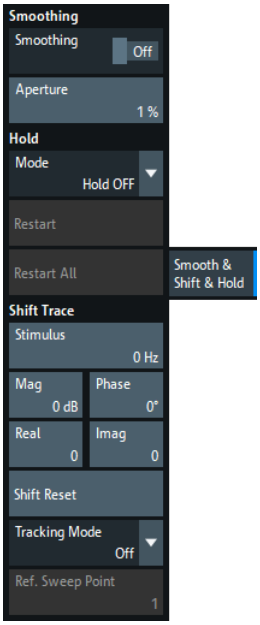
Remote command:

```
CALCulate<Chn>:STATistics:DOMain:USER:SHOW
```

5.5.6 Smooth & Shift & Hold tab

Provides various functions to modify the entire measured trace.

- The analyzer can export the raw complex (unformatted) data or formatted data. The unformatted data are independent of all "Smooth & Shift & Hold" settings; see "Formatted Values" on page 480.
- For complex traces, if marker format and trace format do not coincide, the marker values are calculated before Smoothing and Hold are applied. In this case, the displayed marker values may not behave as expected.



Smoothing
Activates the smoothing function for the active trace, which can be a data or a memory trace.

With smoothing active, the trace value at each sweep point is replaced by the arithmetic mean of the trace values at the sweep point itself, and at neighboring sweep points (symmetrically, to the left and to the right). The number of trace values to be averaged is defined by the Aperture and can be adjusted according to the properties of the trace.

Smoothing operates on the formatted trace and does not significantly increase the measurement time.

Tip: The sweep average is an alternative method of compensating for random effects on the trace by averaging consecutive traces. Compared to smoothing, the sweep average requires a longer measurement time but does not have the drawback of averaging out quick variations of the measured values.

See Section 5.9.4, "Average tab", on page 534.

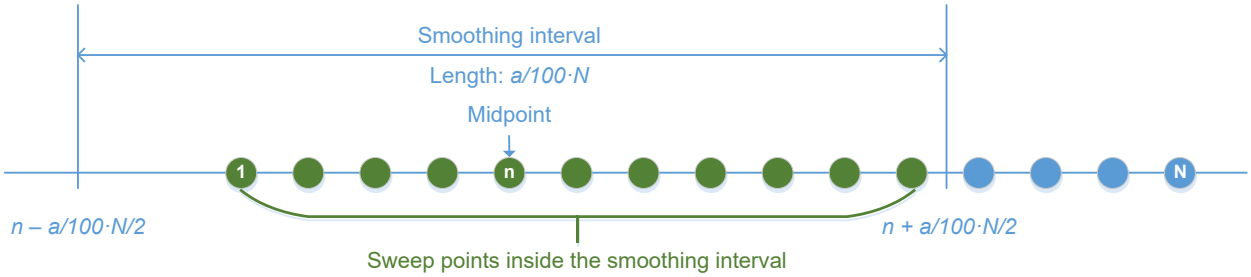
Remote command:
`CALCulate<Chn>:SMOothing[:STATe]`

Aperture
Defines how many trace values are averaged to smooth the trace if Smoothing is switched on.

An "Aperture" of *a* % means that the value at sweep point $n \in \{1, ..., N\}$ is replaced by the arithmetic mean of the values at sweep points

$max\{1, \lceil n - a/100 \cdot N/2 \rceil\}, ..., min\{N, \lfloor n + a/100 \cdot N/2 \rfloor\}$,

where *N* is the number of points of the sweep (or sweep segment for segmented sweeps).



The range definition above takes boder cases and rounding into account.

Tips: Finding the appropriate aperture

A large smoothing aperture enhances the smoothing effect but can hide quick variations of the measured values and thus produce misleading results.

To avoid errors, observe the following recommendations.

- Start with a small aperture and increase it only as long as you are certain that the trace is still correctly reproduced.
- Select a smoothing aperture that is small compared to the width of the observed structures (e.g. the resonance peaks of a filter). If necessary, restrict the sweep range or switch smoothing off to analyze narrow structures.

Remote command:
CALCulate<Chn>:SMOothing:APERture

Hold
Selects the "Max Hold" (peak hold) or "Min Hold" function for the active trace, or disables both functions ("Hold Off"). With enabled "Max Hold" or "Min Hold" function, the displayed trace shows the maximum or minimum values that the analyzer acquired since the start of the measurement. The "Max Hold" and "Min Hold" traces are real; they are based on the magnitude of the trace values (the phase values are discarded).

The "Hold" process can be restarted any time using "Restart" (current trace) or "Restart All" (all traces in the active recall set). The "Hold" process is also restarted automatically when the channel or trace settings are changed so that the previous measurement results are no longer compatible.

Note: A memory trace is unformatted by definition. Therefore, a "to memory" operation on a "Hold" trace actually stores the last measured trace data instead of the current "Max Hold" or "Min Hold" values.

Remote command:
CALCulate<Chn>:PHOLd

Shift Trace
Functions for shifting the active trace in horizontal and vertical direction.

Stimulus ← Shift Trace
Shifts the active trace in horizontal direction, leaving the positions of all markers unchanged. The unit of the offset value depends on the sweep type.

Note:
A "Stimulus" shift can be used in cartesian and in complex diagrams. The visible effect depends on the diagram type:
- In cartesian diagrams, the trace is shifted relative to the markers and the x-axis.
- In complex diagrams, the trace is not affected.

Remote command:
DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:X:OFFSet

Mag / Phase / Real / Imag ← Shift Trace
Modifies the active trace by adding and/or multiplying complex constants.

The trace points are modified according to the following formula:

$$M_{new} = M_{old} \cdot 10^{<\text{Magnitude}>/20\,\text{dB}}\, a \cdot e^{j \cdot <\text{Phase}>/180^\circ} + <\text{Real}> + j <\text{Imag}>$$

The formula and the different constants are adjusted to the different display formats of a trace:
- The "Mag" factor shifts a dB Mag trace in vertical direction, leaving the phase of a complex parameter unchanged.
- The "Phase" factor rotates a trace that is displayed in a polar diagram around the origin, leaving the magnitude unchanged.
- The "Real" value shifts a real trace in vertical direction, leaving the imaginary part unchanged.

- The "Imag" value shifts an imaginary trace in vertical direction, leaving the real part unchanged.

Tip: Shifting the trace by constant values is a simple case of trace mathematics. Use the "User Def Math" dialog to define more complicated mathematical operations (see Section 5.5.4.2, "User Def Math dialog", on page 461).

Remote command:
`DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:Y:OFFSet`

Shift Reset
Resets the static shifting, defined using Stimulus and Mag / Phase / Real / Imag, respectively.

Tracking Mode/Ref Sweep Point
Allows you to take the trace value at a selected sweep point as the reference point for a dynamic shift of the active trace.

With a raw complex trace value $z = Re(z) + j \cdot Im(z) = |z| \, e^{j \cdot \varphi(z)}$ and a raw complex value $zr = Re(zr) + j \cdot Im(zr) = |zr| \, e^{j \cdot \varphi(zr)}$ at the reference point (for the current sweep), the dynamically shifted trace value Dyn_Shift(z) is calculated as follows:

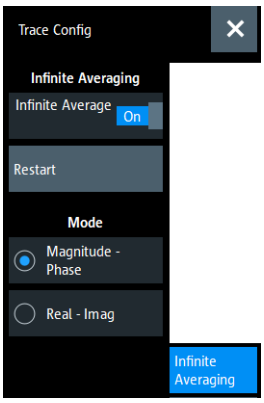
"Amplitude"	$Dyn_Shift(z) = z / \|zr\|$
"Phase"	$Dyn_Shift(z) = z \, e^{-j \cdot \varphi(zr)}$
"Real"	$Dyn_Shift(z) = z - Re(zr)$
"Imag"	$Dyn_Shift(z) = z - Im(zr)$
"Complex"	$Dyn_Shift(z) = z - zr$

Remote command:
`CALCulate<Chn>:TRACking:MODE`
`CALCulate<Chn>:TRACking:SWEep:POINt`

5.5.7 Infinite Averaging tab

The controls on the "Infinite Averaging " tab allow you configure infinite averaging for the active trace.

Infinite averaging means the average at a sweep point is taken over all trace values at this sweep point since the previous restart of the averaging cycle.



Infinite Average/Restart
"Infinite Average" enables or disables "Infinite Averaging " (default: disabled), "Restart" starts a new averaging cycle.

Remote command:
`CALCulate<Chn>:IAVerage[:STATe]`

Mode
Selects the quantities to be averaged.

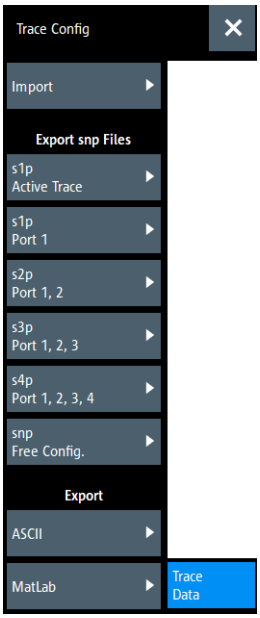
"Magnitude - Phase"	Perform averaging of magnitude and phase (default) of the complex trace value.
Real - Imag	Perform averaging of the real and imaginary part of the complex trace value (i.e. over the complex value itself).

Remote command:
`CALCulate<Chn>:IAVerage:MODE`

5.5.8 Trace Data tab

Stores one or several data or memory traces to a file or loads a memory trace from a file.

Background information
Refer to Section 4.4.2, "Trace files", on page 143.



All buttons on the "Trace Data" tab serve as "openers" for related dialogs:

- "Import" calls up a dialog to load a memory trace from a trace file; see Section 5.5.8.1, "Import Complex Data dialog", on page 476.

- The buttons in the "Export snp Files" section call up a dialog to store data or memory traces to a trace file of the corresponding content and file format; see Section 5.5.8.2, "Export Data - <File Type> dialog", on page 477.
- "snp Free Config" opens a dialog to define the port assignment for the created Touchstone (`*.s<n>p`) file. See Section 5.5.8.4, "Select Ports dialog", on page 482.

Exporting data from the firmware simulation

When working with the firmware simulation R&S ZNXSIM, make sure to deactivate "Simulation Noise" before exporting traces to file. Otherwise the exported results comprise an "artificial" random component.

Since version 3.40 of the analyzer firmware, Simulation Noise can be activated or deactivated from the analyzer GUI.

5.5.8.1 Import Complex Data dialog

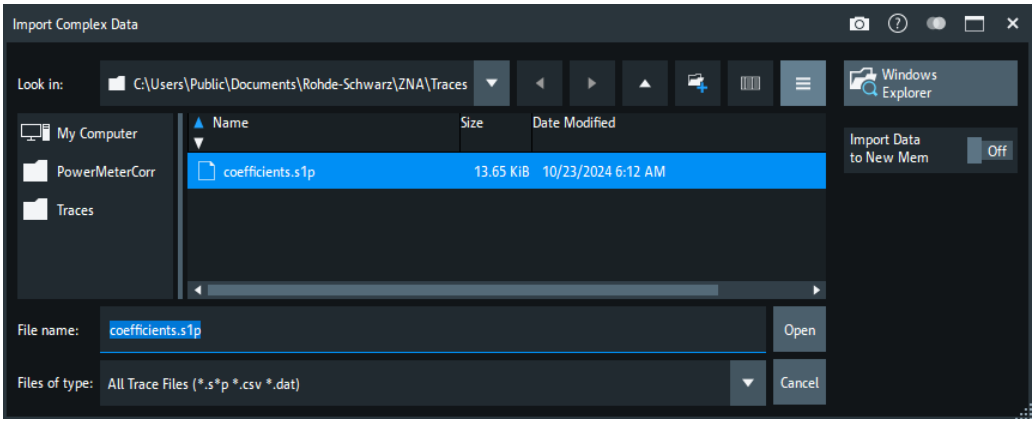
"Import Complex Data"

The "Import Complex Data" dialog is a standard "Open File" dialog with an additional button. It loads a memory trace from a trace file. Trace files are ASCII files with selectable file format. The loaded trace data is used to generate a memory trace, which is coupled to the active data trace.

Access: Trace – [Trace Config] > "Trace Data" > "Import"

Background information

Refer to Section 4.4.2, "Trace files", on page 143.



On loading data from a trace file with several traces, the analyzer displays a dialog to select one or more of the traces stored in the file (see Section 5.5.8.3, "Select Parameter dialog", on page 482). E.g., for an `*.s2p` Touchstone file, the box offers all four 2-port S-parameters (see Section 4.4.2.1, "Touchstone files", on page 144).

Coupling between the imported memory trace and the active data trace implies that the stimulus values of the imported data and of the active trace must be compatible. Com-

patibility means that the "Sweep Type" of the two traces must match; the position and number of the sweep points do not have to be the same.

The analyzer checks for compatibility before importing data. The "Select Parameter" box remains empty if the selected file contains no compatible data.

Since FW V4.25 the R&S ZNB supports UTF8 files with and without BOM for CSV trace data import.

Import Data to New Mem
Specifies whether the loaded data overwrite an existing memory trace, if available (box disabled), or whether they are used to generate a new memory trace (control enabled).

If the box is disabled and the active trace is a memory trace, then this memory trace is overwritten. If the control is disabled and the active trace is a data trace, then the data trace's last created memory trace is overwritten (or a new memory trace is created, in case there was previously no memory trace assigned to this data trace).

Remote command:
`MMEMory:LOAD:TRACe`

5.5.8.2 Export Data - <File Type> dialog

The "Export Data - <File Type>" dialog stores data or memory traces to a trace file. Trace files are ASCII files with selectable file format.

Access:

- Trace – [Trace Config] > "Trace Data" > "Export snp Files" – ...
- Trace – [Trace Config] > "Trace Data" > "Export" – ...

Data export can serve many purposes, e.g.:

- To process and evaluate measurement data in an external application.
- To store measurement data and reimport it in a future measurement session.

Background information
Refer to the following sections:

- Section 4.4.2, "Trace files", on page 143.
- Section 4.4.2.3, "Finding the best file format", on page 152

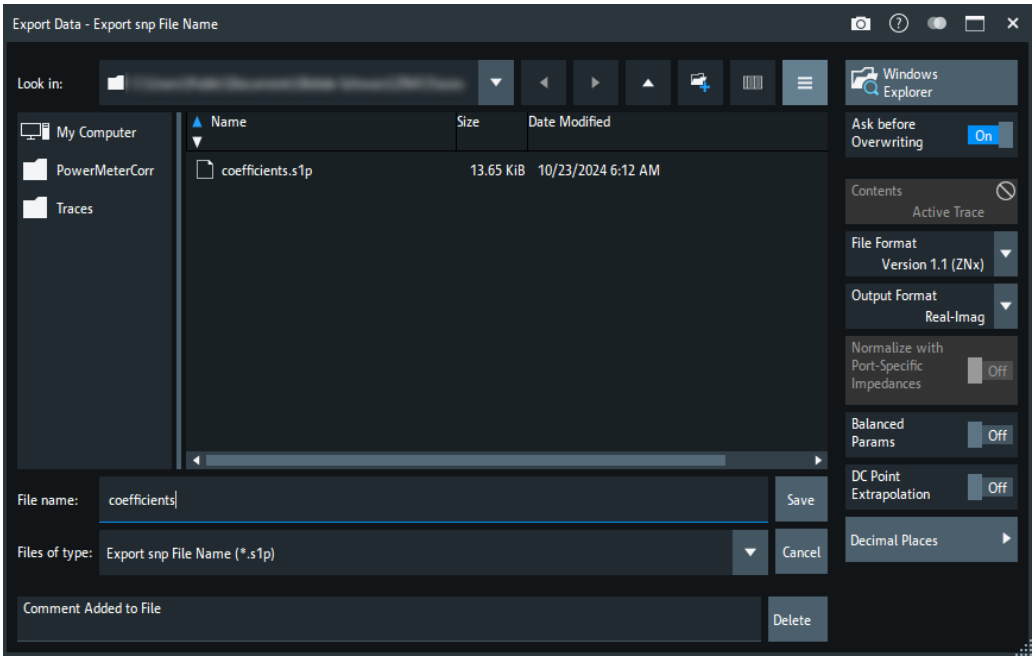


Figure 5-22: Touchstone file export

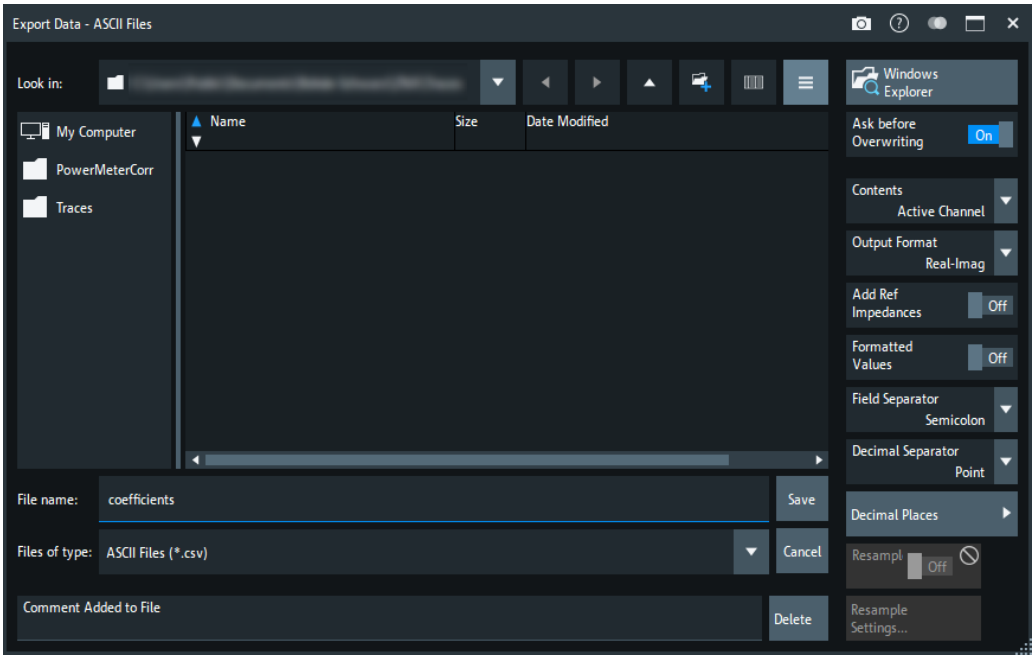


Figure 5-23: ASCII Trace files export

The "Export Data - <File Type>" dialog is a standard "Save File" dialog with several additional buttons to specify the export options. Many options depend on the selected export file format ("Files of type"). The displayed controls change accordingly.

The export options are remembered when the dialog is closed using "Save".

Ask Before Overwriting
Activates a message box to be displayed before an older trace file with the same file-name and directory is overwritten.

Contents
For ASCII (`*.csv`) and MATLAB (`*.dat`) export, you can include the active trace, all traces of the active channel (including memory traces), or all traces in all channels.

For Touchstone file export, it is possible to export the traces of the active channel or of all channels. Consider "Conditions for Touchstone file export" on page 150.

In both cases, the R&S ZNB creates one file per channel.

File Format
Available for Touchstone export only.

Selects the Touchstone file format to be generated. See Section 4.4.2.1, "Touchstone files", on page 144.

Remote command:
`MMEMory:STORe:TRACe:OPTion:FORMat`

Output Format
Selects the format of the exported raw, complex measurement values. The exported values can be represented by the real and imaginary parts, the linear magnitude and phase, or dB magnitude and phase; see also "Formatted Values" on page 480.

Export of Formatted Values is not available for Touchstone files.

Normalize with Port-Specific Impedances
For Touchstone file export only. If enabled, the firmware renormalizes the exported S-parameters according to the port-specific reference impedances instead of a common target impedance of 50 Ω (see "Renormalization of S-parameters" on page 150).

Note however, that during import the firmware always assumes a common reference impedance of 50 Ω.

Balanced Params
For Touchstone file export only: activates/deactivates the export of balanced (and mixed-mode) S-parameters.

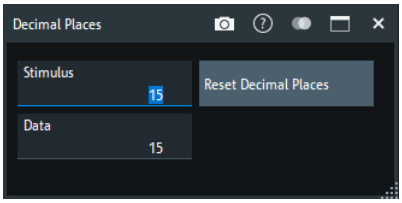
Note:
- Please consider "Conditions for Touchstone file export" on page 150. In particular: if only one port of a balanced port is selected, then the export fails.
- In contrast to single-ended S-parameters, balanced S-parameters are always normalized.

Remote command:
`MMEMory:STORe:TRACe:OPTion:BALanced`

DC Point Extrapolation
For Touchstone file export only: enables/disables extrapolation of trace data to the DC point (0 Hz).

Remote command:
`MMEMory:STORe:TRACe:OPTion:EXTRapolate`

Decimal Places
Opens a dialog that allows you to specify the number of decimal places for stimulus and data values in all supported trace data files (Touchstone, ASCII, MatLab).



The allowed range is 1 (minimum precision) to 15 (maximum precision). "Reset Decimal Places" resets these numbers to their respective defaults, which is 15 in both cases.

Note: In the current implementation of the Touchstone file export, only the "Version 1.1 (ZNx)" File Format file format supports independent settings for stimulus and data values. For formats "Version 1.1" and "Version 2.0", both stimulus and data values are exported with "Data" decimal places.

Remote command:
`MMEMory:STORe:TRACe:OPTion:DECimals:STIMulus`
`MMEMory:STORe:TRACe:OPTion:DECimals:DATA`

Add Ref Impedances
For ASCII (`*.csv`) or MATLAB (`*.dat`) files only: Includes the reference impedances Z_0 for all analyzer ports in the file header.

Formatted Values
For ASCII (`*.csv`) or MATLAB (`*.dat`) files only: Selects the format for the exported trace data.

- **Control disabled (off):** Export the raw complex (unformatted) measurement values, represented by the real and imaginary parts, the linear magnitude and phase, or dB magnitude and phase.
  The exported complex trace values are the values at the beginning of the trace data flow. None of the following stages (trace mathematics, shift, time domain gate, trace formatting and smoothing) affects the exported data. "Save" writes the raw stimulus values (frequency/power/time, according to the sweep type) and the raw, complex measurement points to a file. See Section 4.1.5, "Data flow", on page 89.
  Export of complex data is available for all trace file types.
- **Control enabled (on):** Export the values as they are displayed in the diagram, e.g. export the dB magnitude, if trace format "dB Mag" is selected. The trace file does not necessarily contain the full (complex) information about the trace.
  For trace formats involving Cartesian diagrams ("dB Mag", "Real", "Imag"...), the stimulus value and a single real response value is exported. For circular diagrams, both the real and imaginary part of the response value is exported.
  The trace values are the fully processed values as they appear in the diagram area. They correspond to the results in the marker info field. All possible stages of the trace data flow (e.g. trace formats, trace mathematics, time domain transform, shift, smoothing) are considered. Some trace functions (e.g. time scale, shift stimulus) also affect the stimulus values.
  Export of formatted data is not available for Touchstone files.

Field Separator
For ASCII (`*.csv`) file export, this property defines the separator the analyzer uses to separate different numbers in each line of the file. For MATLAB (`*.dat`) export, always "Comma" is used.

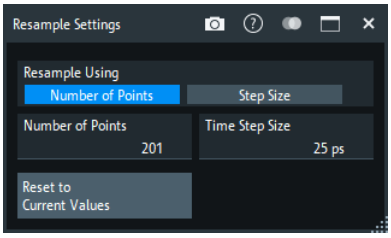
Decimal Separator
For ASCII (`*.csv`) files only: Selects either the "Point" or the "Comma" (if needed to process the exported data with an external application) as a separator for decimal numbers.

Resample/Resample Settings
Available for ASCII (`*.csv`) file export only.

"Resample" is enabled if:

- The instrument is equipped with the time domain option.
- The active trace is represented in the Time Domain.
- Only the active trace shall be exported.
- The active trace shall be exported as formatted values.

In this case, "Resample Settings" opens a dialog that allows you to select either the "Number of points" or the "Time Step Size":



The "Number of Points" can be selected between 1 and the Number of Points N_f of the underlying frequency sweep (default).

The upper limit for "Time Step Size" is the Time Span of the time sweep, lower limit (and default) is this time span divided by N_f-1.

"Reset to Current Value" resets the number of points to the "Number of Points" of the sweep and adjusts the "Time Step size" accordingly.

Remote command:
```
MMEMory:STORe:TRACe:OPTion:RSAMple
MMEMory:STORe:TRACe:OPTion:RSAMple:POINts
MMEMory:STORe:TRACe:OPTion:RSAMple:TIMStep
MMEMory:STORe:TRACe:OPTion:RSAMple:PRESet
```

Save
Stores the trace data, according to the selected options.

Tip: Note the conditions described in "Conditions for Touchstone file export" on page 150.

Remote command:
```
MMEMory:STORe:TRACe:PORTs
MMEMory:STORe:TRACe
MMEMory:STORe:TRACe:CHANnel
```

Comment Added to File
Defines a comment to be added to the exported trace data file.

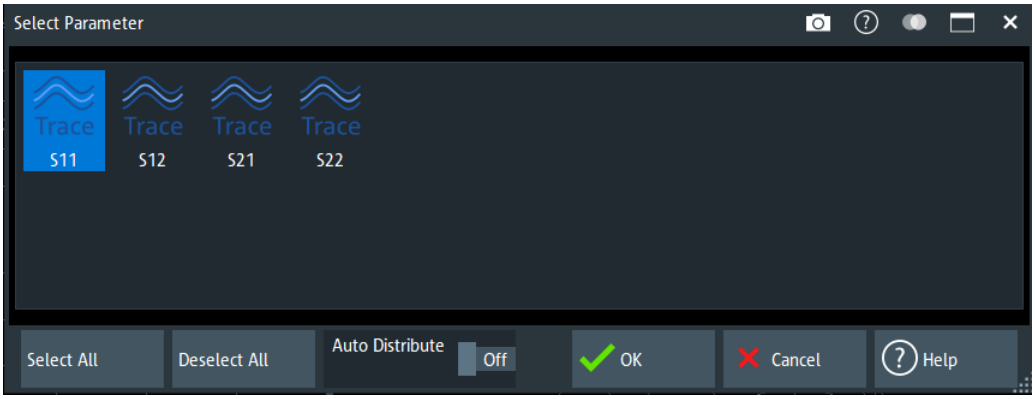
If empty (←"Delete"), no comment is added.

Remote command:
`MMEMory:STORe:TRACe:OPTion:COMMent`

5.5.8.3 Select Parameter dialog

The "Select Parameter" dialog provides a selection of measurement results (e.g. S-parameters) or traces, e.g. for trace import, import of power correction coefficients, limit line import.

Access: The dialog may be called from several dialogs, for example on pressing "Open" in the Import Complex Data dialog.



Select All / Deselect All
During trace data import, selects/deselects all traces contained in the opened trace file.

Auto Distribute
Available for trace data import only.

If enabled, a selected trace S_{ij} is imported as a memory trace for all data traces in the current recall set that are measuring S_{ij}.

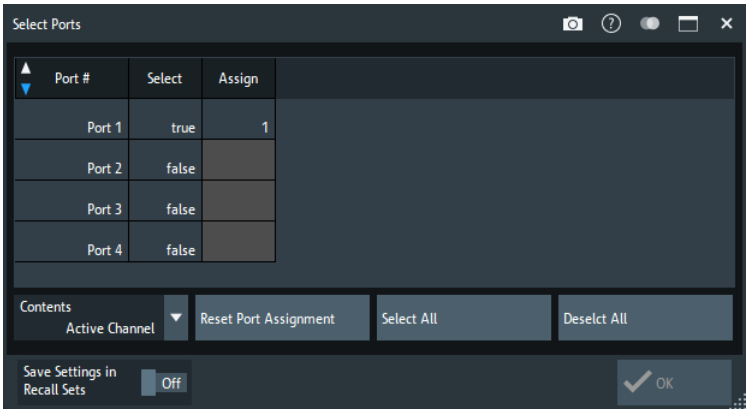
Remote command:
`MMEMory:LOAD:TRACe:AUTO`

5.5.8.4 Select Ports dialog

The "Select Ports" dialog defines the port assignment for the created Touchstone (`*.s<n>p`) file.

Access: Trace – [Trace Config] > "Trace Data" > "snp Free Config"

Touchstone files and file export
Note the conditions described in "Conditions for Touchstone file export" on page 150.



Checks and Messages in the Dialog

After each port or channel selection, the R&S ZNB checks the channel data for compatibility with the trace export conditions. If data from "All Channels" are exported, every channel must contain a compatible set of traces; see "Conditions for Touchstone file export" on page 150.

The "OK" button is available only if no error message is displayed in the dialog.

Select / Select All / Deselect All

Selects the ports to be considered for the S-parameter export.

Example: With ports 1 and 2 selected, S-parameters S_{11}, S_{12}, S_{21} and S_{22} are exported.

Remote command:

`MMEMory:STORe:TRACe:PORTs`

Assign

Selects the port number assignment in the created `*.s<n>p` file. By default, analyzer and `*.s<n>p` port numbers are identical. You can interchange the port assignment to change the order of the S-parameters in the created ´Touchstone file. Each of the analyzer port numbers must be assigned to one `*.s<n>p` port number.

Active Channel / All Channels

Selects data export for the active channel or for all channels.

Reset Port Assignments

Restores the identity between original and assigned port number.

Save Settings in Recall Set

The "Select Ports" dialog preserves the selected and assigned port numbers. By default (disabled), these settings are global, i.e. they are identical for each recall set and are not affected by a [Preset]. However you can also choose to preserve specific settings for the current recall set (enabled).

Note however that the settings are not persisted unless the "Select Ports" dialog is closed using "OK".

5.6 Lines softtool

The "Lines" softtool allows you to define limits for the measurement results, visualize them in the diagrams and activate/deactivate the limit check. The analyzer provides upper, lower, ripple and circle limits. In addition, the "Lines" softtool provides functions to limit complex diagrams to a user-defined "Display Circle" and to add user-defined horizontal lines to cartesian diagrams.

Access: Trace – [Line] hardkey

Background information

Refer to Section 4.4.1, "Limit check", on page 135.

5.6.1 Limit Test tab

Defines limit lines for the measurement results (upper and lower limits), visualizes them in the diagrams and enables/disables the limit check.

Limit lines are available for all Cartesian diagram types; "dB Mag" limits can also be checked in complex diagrams (Smith, Polar).

Background information

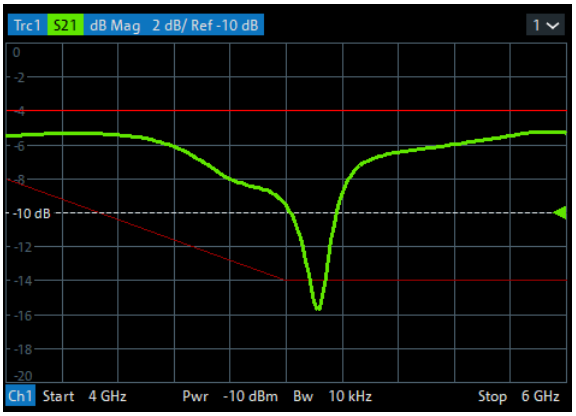
Refer to Section 4.4.1, "Limit check", on page 135.

5.6.1.1 Controls on the Limit Test tab



The "Define Limit Line" button opens the "Define Limit Lines" dialog (see Section 5.6.1.2, "Define Limit Lines dialog", on page 488).

Show Limit Line
Shows or hides the limit line associated with the active trace in a Cartesian diagram area.



The limit line colors are defined in the Define User Color Scheme dialog (System > [Display] > "Config" > "Define User Color"). You can choose between various options:

- Display upper and lower limit lines with different colors.
- Assign the same color to traces and associated limit lines.
- Assign different colors to limit line segments with disabled limit check.

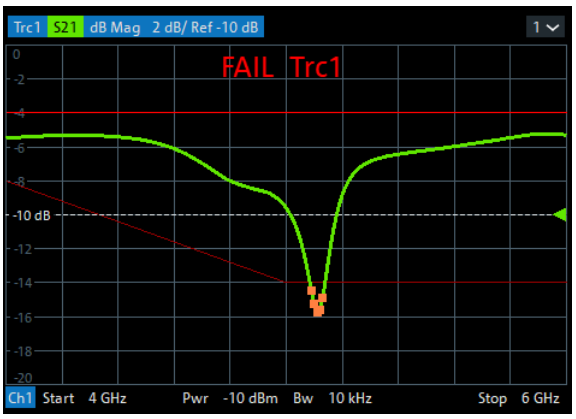
Note: Displaying the limit line and limit check are independent of each other: Hiding the limit line does not switch off the limit check.

Remote command:
`CALCulate<Chn>:LIMit:DISPlay[:STATe]`

Limit Check
Enables/disables the limit check for the active trace. If enabled, an additional info field is displayed in the diagram, indicating the "PASS" or "FAIL" state.

Limit violations are marked with squares, whose color is defined in the Define User Color Scheme dialog ("Limit Fail Trace Color").

A TTL signal indicating pass or fail can be generated in addition.



The "Limit Fail Trace Color" and the appearance of the limit fail symbols are defined in the Define User Color Scheme dialog ("Display" > "Config" > "Define User Color").

Note:

- Limit check and display of limit lines are independent of each other:
  – The limit lines can be displayed, no matter if the limit check is enabled.
  – If "Limit Check" is enabled, the limits are checked, no matter if the limit lines are displayed.
  – The limit check can even be enabled, if no limit lines are defined. In this case, the info field displays "No limit defined!" and the limit check always passes.
- Limit lines are defined for a particular trace format. However, the limit check is performed irrespective of the current trace format. The info field indicates the correct "PASS"/"FAIL" state and limit violations are visualized on the trace (if any).

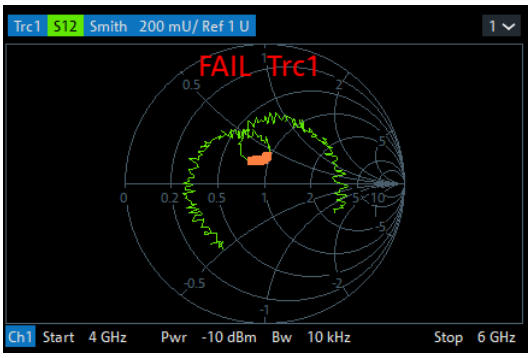


Figure 5-24: Limit line violations in complex trace formats

Remote command:

```
CALCulate<Chn>:LIMit:STATe
CALCulate<Chn>:LIMit:LOWer:STATe
CALCulate<Chn>:LIMit:UPPer:STATe
CALCulate<Chn>:LIMit:FAIL?
CALCulate:LIMit:FAIL:ALL?
CALCulate<Chn>:LIMit:STATe:AREA
CALCulate:LIMit:POINts:LOWer?
CALCulate:LIMit:POINts:UPPer?
```

Clear Test

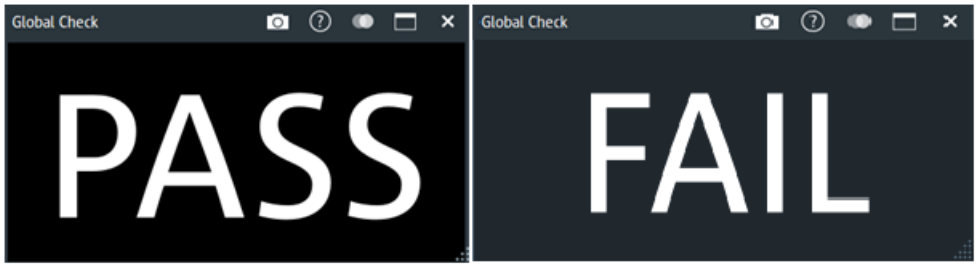
Resets the limit check results.

Remote command:

```
CALCulate<Chn>:LIMit:CLEar
```

Global Check

Enables or disables the global limit check including upper/lower limits and ripple limits. The global limit check is a composite limit check over all traces of the current recall set. The result of the global check appears in a popup box whenever "Global Check" is selected.



- "PASS" represents pass for all traces with enabled limit check. A trace without limit lines or with disabled individual limit check always passes the global check.
- "FAIL" means that the limit check for one or more traces failed.

Remote command:
`CALCulate:CLIMits[:STATe]`
`CALCulate:CLIMits:FAIL?`

TTL1 Pass / TTL2 Pass
Assigns the active trace to the low-voltage (3.3 V) TTL output signals at the USER PORT (see Section 13.3.1.1, "USER PORT", on page 1702).

Monitoring a single trace

If "TTL1 Pass" ("TTL2 Pass") is selected and the trace passes all active limit checks, then the TTL signal is applied to pin 13 (pin 14) of the USER PORT. If one of the limit checks fails, then no TTL signal is generated.

Monitoring several traces

If a channel contains several traces, it is possible to assign each of them to any TTL output. The assignment divides the traces of the channel into four groups:

- Not assigned to signal 1 or signal 2
- Assigned to signal 1, but not to signal 2
- Assigned to signal 2, but not to signal 1
- Assigned to both signals

If several traces are assigned to a pass/fail signal, then the TTL signal is only generated if all traces are within their respective limits. It is switched off if one trace exceeds those limits.

Application: Graduated quality check

The two pass/fail signals can be used to distinguish three quality levels of a DUT. The test is performed on two identical traces Trc1 and Trc2 within the same channel. Trc1 is configured with a tighter, Trc2 with a less restrictive set of limit lines. For Trc1 "TTL1 Pass" is enabled, for Trc2 "TTL2 Pass".

- TTL1: signal
  If Trc1 passes (and so does Trc2), the quality of the DUT is good.
- TTL1: no signal, TTL2: signal
  If Trc1 fails but Trc2 passes, the quality of the DUT is still sufficient.
- TTL1: no signal, TTL2: no signal
  If both Trc1 and Trc2 fail, the quality is poor.

Instead of using two traces, it is possible to consider two groups of traces that are assigned to "TTL1 Pass" and "TTL2 Pass", respectively.

Note: Default values "TTL1 Pass" / "TTL2 Pass" for new traces can be set in the "Advanced" tab of the "System Config" dialog (see "TTL Pass Default Values" on page 820).

Remote command:
`CALCulate<Chn>:LIMit:TTLout<Pt>[:STATe]`

Shift Lines
The "Stimulus" and "Response" values shift all limit lines in x and y direction, respectively, without modifying the related line segments.

Remote command:
`CALCulate<Chn>:LIMit:X:OFFSet`
`CALCulate<Chn>:LIMit:Y:OFFSet`

5.6.1.2 Define Limit Lines dialog

The "Define Limit Lines" dialog defines the limit lines for the active trace on a segment-by-segment basis. For each segment, the limit is defined as a line connecting two points (with linear or logarithmic "Interpolation"), or as a user-defined formula.

Access: Trace – [Line] > "Limit Test" > "Define Limit Line..."

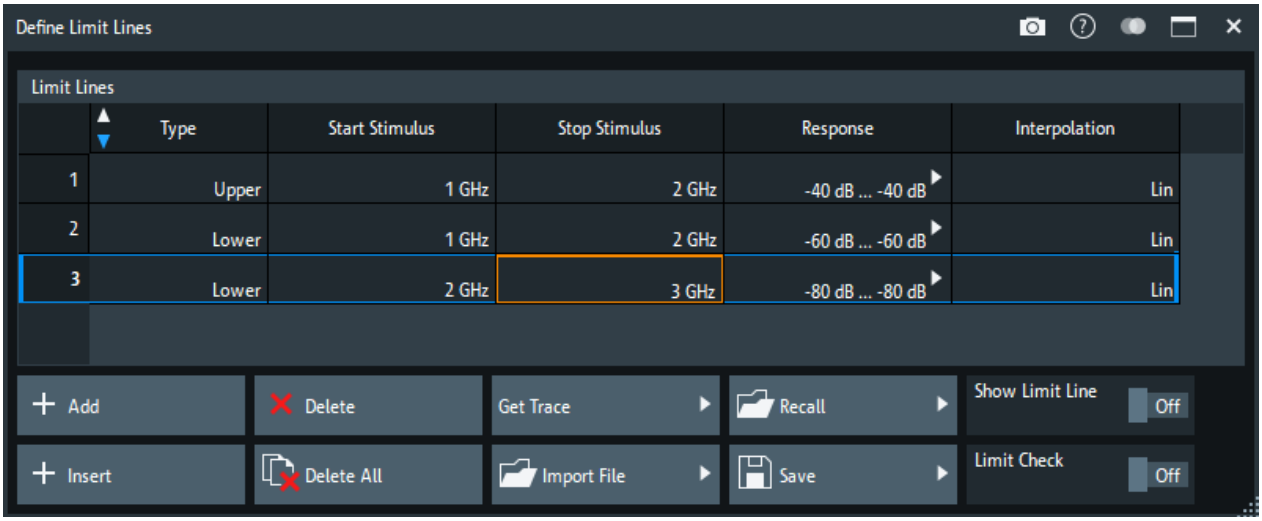
Background information
Refer to Section 4.4.1, "Limit check", on page 135.

When you switch to a different sweep type, existing limit lines are preserved. With a different sweep variable, however, the limit lines cannot be displayed and the corresponding limit check cannot be executed.

If you want to define limit lines for the new sweep variable, you first have to delete all existing limit lines for the old sweep variable.

Controls on the Define Limit Lines dialog

The "Define Limit Lines" dialog contains a table to edit the individual segments of the limit lines. The buttons below the table extend or shorten the segment list.



Creating limit lines with minimum effort

Choose one of the following methods to create and handle limit lines efficiently:

- To define limit lines with only a few segments, select "Add" and edit each segment in the Segment List individually.
- Select a data or memory trace as a limit line ("Import Trace...") or import a trace stored in a file ("Import File...").
- Save your limit lines to a file so you can reuse or modify them later sessions ("Save Limit Line...", "Recall Limit Line...").

Segment List

Defines the individual limit line segments.

The table contains an automatically assigned current number for each segment plus the following editable columns:

- "Type" indicates whether the segment belongs to an "Upper" or a "Lower" limit line, or if the limit check at the segment is switched "Off". Switching off the limit check does not delete the segment but changes its screen color.
- "Start Stimulus" is the stimulus (x-axis) value of the first point of the segment.
- "Stop Stimulus" is the stimulus (x-axis) value of the last point of the segment.
- "Response"
  - For a limit line segment that is defined as linear functions of the stimulus, "Response" displays the y-axis values at the start and stop stimulus (x-axis) values.
  - For formula-defined line segments, it displays the formula.

  Tap the "Response" field to open the Response dialog that offers the full range of configuration possibilities.
- "Interpolation" determines whether the limit line segment is interpolated linearly or logarithmically. The latter is possibly more suitable for logarithmic sweeps.

Remote command:

```
CALCulate<Chn>:LIMit:SEGMent:COUNt?
CALCulate<Chn>:LIMit:SEGMent<Seg>:TYPE
CALCulate<Chn>:LIMit:SEGMent<Seg>:STIMulus:STARt
```

```
CALCulate<Chn>:LIMit:SEGMent<Seg>:STIMulus:STOP
CALCulate<Chn>:LIMit:SEGMent<Seg>:AMPLitude:STARt
CALCulate<Chn>:LIMit:SEGMent<Seg>:AMPLitude:STOP
CALCulate<Chn>:LIMit:CONTrol[:DATA]
CALCulate<Chn>:LIMit:CONTrol:SHIFt
CALCulate<Chn>:LIMit:DATA
CALCulate<Chn>:LIMit:LOWer[:DATA]
CALCulate<Chn>:LIMit:LOWer:SHIFt
CALCulate<Chn>:LIMit:UPPer[:DATA]
CALCulate<Chn>:LIMit:UPPer:FEED
CALCulate<Chn>:LIMit:SEGMent<Seg>:INTerpol
```

Add / Insert / Delete / Delete All

The first four buttons below the segment list extend or shorten the list. The analyzer places no restriction on the number of segments in a limit line.

- "Add" adds a new segment to the end of the list. The new segment extends from the "Stop Stimulus" value of the last segment to the end of the sweep range. Its response values are equal to the "Stop Response" value of the last segment.
- "Insert" adds a new segment before the active segment (marked by a blue background in the first column of the segment list). The new segment extends from the "Stop Stimulus" value of the segment before the active segment to the "Start Stimulus" value of the active segment. Its response values are equal to the "Start Response" value of the active segment. The segment numbers in the list are adapted.
  If no segment is active, "Insert" is equivalent to "Add".
- "Delete" removes the selected segment from the list.
- "Delete All" clears the entire segment list so it is possible to define or load a new limit line.

Remote command:

```
CALCulate<Chn>:LIMit:DELete:ALL
```

Recall... / Save...

The buttons open an Open File" or "Save File" dialog, which allows you to load a limit line from a limit line file or store the current limit line configuration to a limit line file.

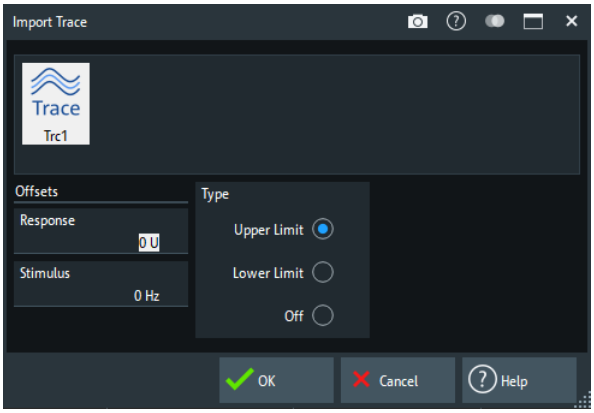
A limit line file (`*.limit`) is a text file whose content describes a limit line configuration. See Section 4.4.1.4, "File format for limit lines", on page 142.

Remote command:

```
MMEMory:LOAD:LIMit
MMEMory:STORe:LIMit
```

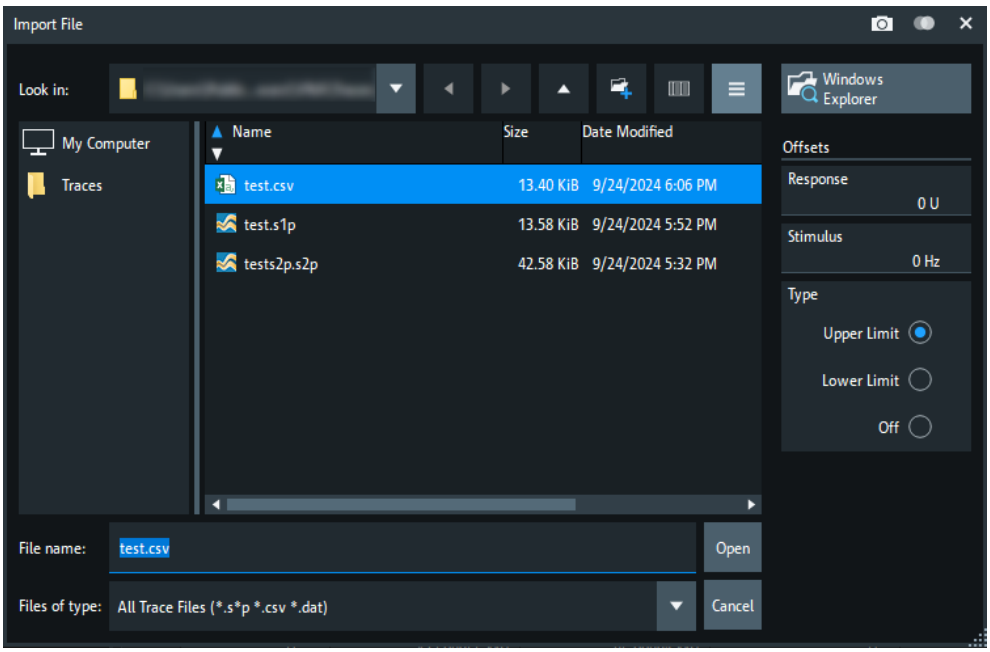
Get Trace... / Import File...

- "Get Trace..." opens a dialog to load a limit line from a data or memory trace in the active recall set.

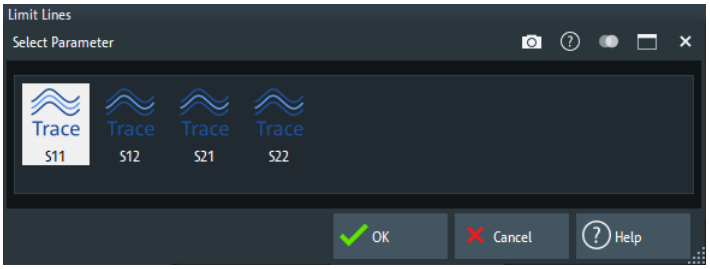


The active trace must be cartesian and the "Format" of the imported trace must be the same as the "Format" of the active trace.

- "Import File..." opens a dialog to load a limit line from a trace file (see Section 5.5.8, "Trace Data tab", on page 475).



In case the selected file contains more than one trace, another popup dialog lets you select the adequate one:



Imported traces are polygonal curves with n points and n – 1, where n is the "Number of Points" of the imported trace (seeSection 5.10.1, "Sweep Params tab", on page 536). The n – 1 segments are appended to the current segment table for further editing. Existing limit line segments are not overwritten.

Both import dialogs contain the following file import settings:

- "Offsets" contains two input fields to define constant offset values for all imported segments. The "Response" offset shifts all segments in vertical direction, the "Stimulus" offset shifts them in horizontal direction. The offsets are added to the start and stop values of all segments.
- "Type" defines whether the imported segments belong to the "Upper" or "Lower" limit line. A third option is to import the segments but disable the limit check ("Off").

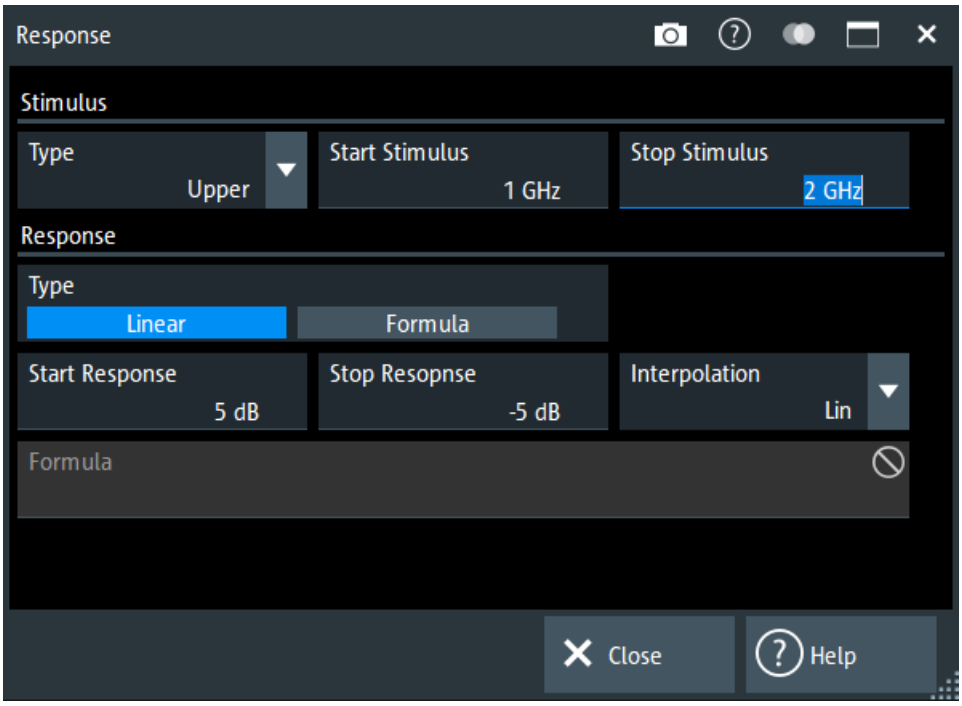
Remote command:

```
CALCulate<Chn>:LIMit:LOWer:FEED
CALCulate<Chn>:LIMit:UPPer:FEED
```

Response dialog

The "Response" dialog allows you to configure the limit line segment in detail.

Access: Define Limit Lines dialog > "Response" cells



For convenience, "Type", "Start Stimulus", "Stop Stimulus", and "Interpolation" are repeated from the Segment List of the "Define Limit Lines" dialog.

Linear/Formula
Allows you to decide how to define the line segment.

- "Linear": Define the line segment as a straight line, connecting the endpoints ("Start Stimulus","Start Response") and ("Stop Stimulus","Stop Response").
  In this case, you can also decide whether linear or logarithmic interpolation is used.

- "Formula": Define the line segment by a custom formula.
  You can enter the formula in the text field to the right of the "Formula" button, or Tap and hold the text field to open the User Def Math dialog.

Remote command:
`CALCulate<Chn>:LIMit:SEGMent<Seg>:FORMula:STATe`
`CALCulate<Chn>:LIMit:SEGMent<Seg>:FORMula`

5.6.2 Ripple Test tab

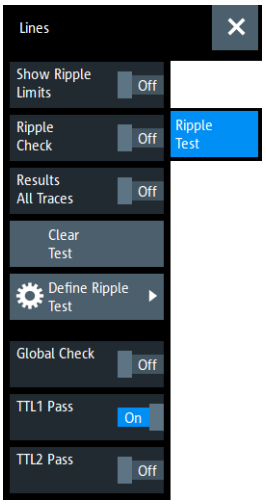
Defines ripple limits for the measurement results, visualizes them in the diagrams and activates/deactivates the ripple limit check.

A ripple test is a special type of limit test where the **difference** between response values in certain stimulus ranges must not exceed configurable limits (ripple limits).

Ripple limits can be defined for cartesian trace formats only and are limited to the trace format they were configured for. If another format is selected, the ripple limit lines are hidden and the limit check is temporarily disabled.

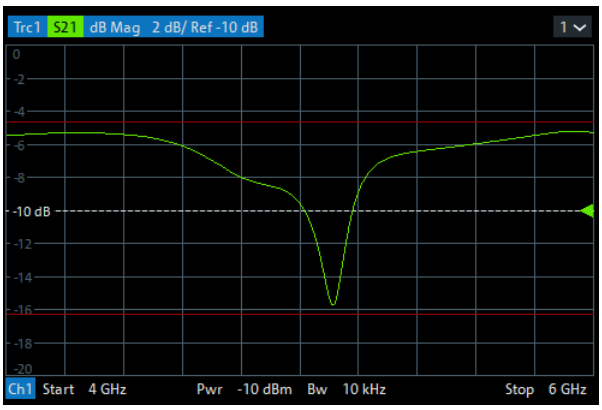
Background information
Refer to Section 4.4.1, "Limit check", on page 135.

5.6.2.1 Controls on the Ripple Test tab



The "Def. Ripple Test" button opens the "Define Ripple Test" dialog (see Section 5.6.2.2, "Define Ripple Test dialog", on page 495).

Show Ripple Limits
Shows or hides the ripple limit lines associated with the active trace in a Cartesian diagram area. The vertical positions of the ripple lines are recalculated after each sweep; only their stimulus range and distance (the ripple limit) are fixed.



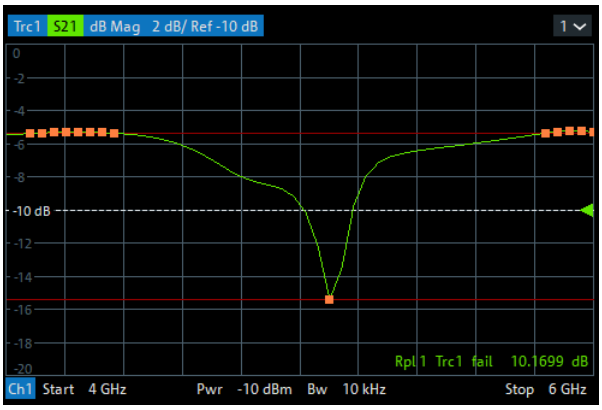
Note: Limit line display and limit check are independent of each other: Hiding the limit line does not switch off the limit check.

Remote command:
`CALCulate<Chn>:RIPPle:DISPlay[:STATe]`

Ripple Check
Switches the ripple limit check of the active trace on or off. When the limit check is switched on, a movable info field shows the pass/fail information and the measured ripple in each ripple limit range.

Limit violations are marked with squares, whose color is defined in the Define User Color Scheme dialog ("Limit Fail Trace Color").

A TTL signal indicating pass or fail can be generated in addition.



Note:
- Ripple check and display of limit lines are independent of each other:
  - The ripple limits can be displayed, no matter if the ripple check is enabled.
  - If "Ripple Check" is enabled, the ripple limits are checked, no matter if they are displayed.
  - The ripple check can even be enabled, if no limit lines are defined. In this case, the info field displays "No ripple defined!" and the limit check always passes.
- For each trace, ripple limits can only be set for a single cartesian trace format. If another trace format is selected, the ripple limit lines are hidden and the ripple check is suspended.

Remote command:
```
CALCulate<Chn>:RIPPle:STATe
CALCulate<Chn>:RIPPle:FAIL?
CALCulate:RIPPle:FAIL:ALL?
CALCulate<Chn>:RIPPle:SEGMent<Seg>:RESult?
CALCulate<Chn>:RIPPle:STATe:AREA
```

Show Results All Traces
Defines the visibility of ripple info fields in the active recall set.

- If disabled, only the ripple info field of the active trace is displayed (in case it has ripple check enabled).
- If enabled, ripple info fields are displayed for all traces with ripple check enabled.

Remote command:
```
CALCulate:RIPPle:DISPlay:RESult:ALL[:STATe]
```

Clear Test
Resets the limit check results.

Remote command:
```
CALCulate<Chn>:RIPPle:CLEar
```

Global Check
See "Global Check" on page 486.

TTL1 Pass / TTL2 Pass
See "TTL1 Pass / TTL2 Pass" on page 487.

5.6.2.2 Define Ripple Test dialog

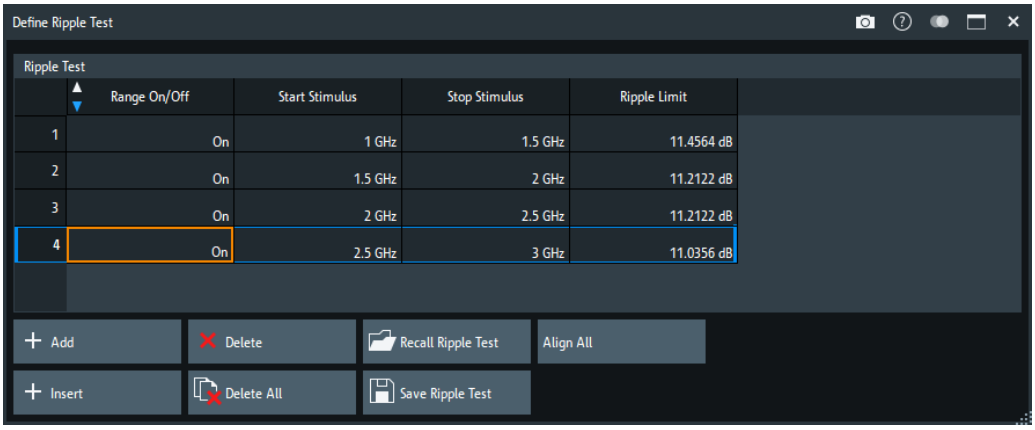
The "Define Ripple Test" dialog defines the ripple limits for the active trace on a range-by-range basis. A separate ripple limit can be assigned to each range.

Access: Trace – [Line] > "Ripple Test" > "Def. Ripple Test..."

Defining ripple limits with minimum effort
Choose one of the following methods to create and handle ripple limit ranges efficiently:

- To configure a limit test with only a few ranges, use "Add" and edit each range in the table individually.
- Use the "Align All" button to create non-overlapping, contiguous ranges of equal width.
- Save your ripple ranges to a file so you can reuse or modify them in later sessions ("Save Ripple Test..., Recall Ripple Test...").

Background information
Refer to Section 4.4.1, "Limit check", on page 135.



The "Define Ripple Test" dialog contains a table to edit the individual ranges of the ripple check ranges. The buttons below the table extend, shorten, or reorder the range list and save/recall ripple test data.

Range List
Defines the individual ripple limit ranges.

The table contains an automatically assigned current number for each range plus the following editable columns:

- "Range On/Off" enables or disables the ripple limit check in each range. Disabling the ripple limit check does not delete the range but hides the entry in the info field.
- "Start Stimulus" is the smallest stimulus (x-axis) value of the range.
- "Stop Stimulus" is the largest stimulus (x-axis) value of the range.
- "Ripple Limit" is the maximum allowed difference between the largest and the smallest trace value in the range.

The ripple limit range is displayed as two parallel, horizontal lines in the diagram. "Stop Stimulus" - "Start Stimulus" is the length of both lines (if the range is within the sweep range); "Ripple Limit" is their vertical distance. See Section 4.4.1.2, "Rules for ripple test definition", on page 138.

Remote command:
```
CALCulate<Chn>:RIPPle:SEGMent<Seg>[:STATe]
CALCulate<Chn>:RIPPle:SEGMent<Seg>:STIMulus:STARt
CALCulate<Chn>:RIPPle:SEGMent<Seg>:STIMulus:STOP
CALCulate<Chn>:RIPPle:SEGMent<Seg>:LIMit
CALCulate<Chn>:RIPPle:SEGMent:COUNt?
```

Add / Insert / Delete / Delete All / Align All
The first four buttons below the range list extend, shorten, or reorder the list.

- "Add" adds a new range to the list. The new range is inserted after the previously selected range. The current range numbers are adapted; the start and stop stimulus values are set so that an overlap is avoided. Moreover, the ripple limit is estimated according to the measured ripple of the trace in the created range. The analyzer places no restriction on the number of ranges assigned to each trace.
- "Insert" adds a new range before the active range (marked by a blue background in the first column of the range list). The new range extends from the "Stop Stimulus" value of the range before the active range to the "Start Stimulus" value of the active

range. Its ripple limit is estimated according to the measured ripple of the trace in the created range. The range numbers in the list are adapted.
If no range is active, "Insert" is equivalent to "Add".

- "Delete" removes the selected range from the list.
- "Delete All" clears the entire range list so it is possible to define or load a new ripple limit line.
- "Align All" redefines existing sweep ranges such that they cover the overall sweep range and have (almost) equal width. The ripple limits are estimated according to the measured ripple of the trace in the created ranges.

Remote command:
`CALCulate<Chn>:RIPPle:CONTrol:DOMain`
`CALCulate<Chn>:RIPPle:DATA`
`CALCulate<Chn>:RIPPle:DELete:ALL`

Recall Ripple Test.../Save Ripple Test...
The buttons open an Open/Save File dialog to load a ripple limit line from a ripple limit file or store the current ripple limit configuration to a file.

Ripple limit files are ASCII files with the default extension `*.ripple` and a special file format. See Section 4.4.1.5, "File format for ripple limits", on page 143.

Remote command:
`MMEMory:LOAD:RIPPle`
`MMEMory:STORe:RIPPle`

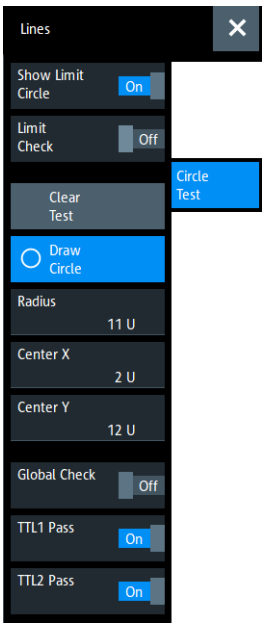
5.6.3 Circle Test tab

Defines circular limit lines for complex trace formats ("Polar", "Smith", "Inv Smith"), visualizes them in the diagram and activates/deactivates the circle limit check.

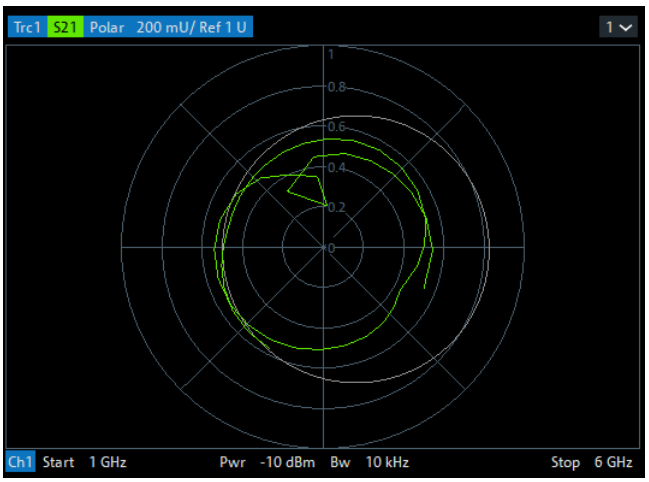
Most of the control elements in the "Circle Test" tab are disabled if the active trace has a cartesian format.



Background information
Refer to Section 4.4.1.3, "Circle limits", on page 139.



Show Limit Circle
Shows or hides the limit line associated with the active trace in a polar diagram area.



The limit line colors are defined in the "Define User Color Scheme" dialog (see Section 5.16.3.2, "Define User Color Scheme dialog", on page 803). You can choose between various options:

- Assign the same color to traces and associated limit lines.
- Assign different colors to limit line segments with disabled limit check.

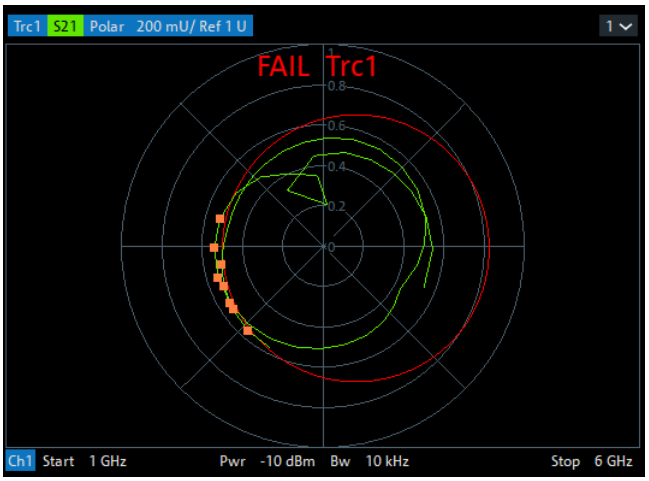
Note: Limit line display and limit check are independent of each other: Hiding the limit line does not switch off the limit check.

Remote command:
`CALCulate<Chn>:LIMit:CIRCle:DISPlay[:STATe]`

Limit Check
Switches the limit check of the active trace on or off. When the limit check is switched on, a movable "PASS" or "FAIL" message is displayed in the diagram.

Limit violations are marked with squares, whose color is defined in the Define User Color Scheme dialog ("Limit Fail Trace Color").

A TTL signal indicating pass or fail can be generated in addition.



The appearance of the limit fail symbols is defined in the "Define User Color Scheme" dialog (see Section 5.16.3.2, "Define User Color Scheme dialog", on page 803). You can choose between various options:

- Change the trace color between failed measurement points.
- Show or hide the colored squares.

Note:

- Circle limit check and display of limit circles are independent of each other:
  - The limit circles can be displayed, no matter if the circle limit check is enabled.
  - If "Limit Check" is enabled, the ripple limits are checked, no matter if they are displayed.
- If result evaluation is limited to a user-defined display circle, the Section 5.6.3, "Circle Test tab", on page 497 is only performed inside this display circle.
- The circle limits can only be checked if the trace format is complex. While a cartesian format is active, the limit circles are hidden and the circle limit check is suspended.
- If the limit check for (cartesian) limit lines is enabled, it is also evaluated in complex diagrams.

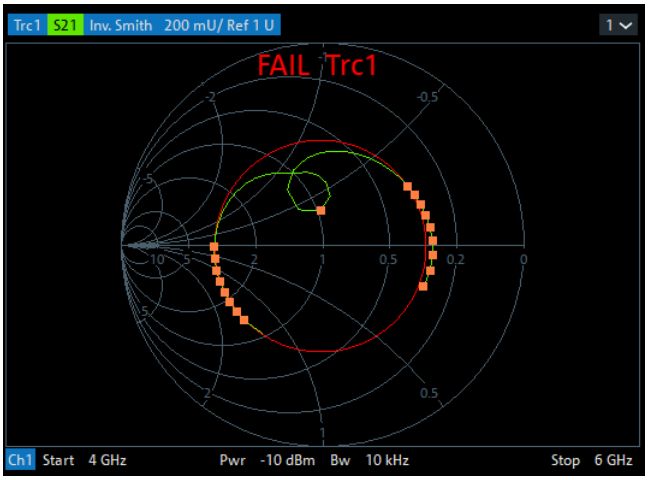


Figure 5-25: Simultaneous dB Mag limit line and circle check

Remote command:
`CALCulate<Chn>:LIMit:CIRCle[:STATe]`
`CALCulate<Chn>:LIMit:CIRCle:FAIL?`
`CALCulate:LIMit:CIRCle:FAIL:ALL?`

Clear Test
Resets the limit check results.

Remote command:
`CALCulate<Chn>:LIMit:CIRCle:CLEar`

Draw Circle
Activates touchscreen or mouse operation; tap the diagram at one border of the limit circle and draw the circle to the required size and position.

Remote command:
n/a

Radius / Center X / Center Y
Defines the limit circle by its radius and its center on the X-axis and Y-axis.

Remote command:
`CALCulate<Chn>:LIMit:CIRCle:DATA`

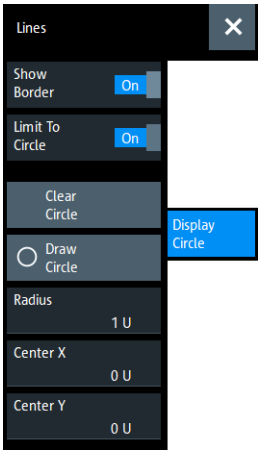
Global Check
See "Global Check" on page 486.

TTL1 Pass / TTL2 Pass
See "TTL1 Pass / TTL2 Pass" on page 487.

5.6.4 Display Circle tab

The "Display Circle" functionality allows you to limit results in complex trace formats (Smith, Polar) to a user-defined circle. In particular, while the trace format is complex, line and circle limit checks are only performed inside the display circle.

Most of the controls on this tab are only active, if the active trace is displayed in a complex format.

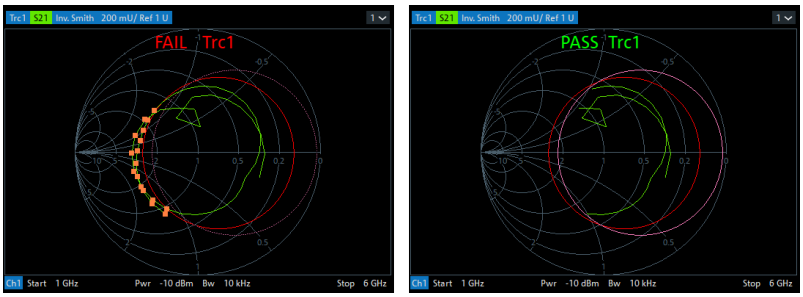


Show Border
If enabled, the border of the Display Circle is shown whenever the related trace is displayed in complex format.

The border color can be modified by a user-defined color scheme (Element "Vertical Range Lines").

Remote command:
`CALCulate<Chn>:LIMit:DCIRcle:DISPlay[:STATe]`

Limit to Circle On/Off
If enabled, only trace points within the configured "Display Circle" are shown, whenever the related trace is displayed in complex format. At the same time, the active limit tests (line, circle) are restricted to the configured "Display Circle".



left ("FAIL") = "Circle Test" enabled, but not limited to "Display Circle"
right ("PASS") = "Circle Test" enabled, but limited to "Display Circle"

Remote command:
CALCulate<Chn>:LIMit:DCIRcle[:STATe]

Clear Circle
Resets the "Display Circle" to its default configuration (unit circle; show border: off; limit to circle: off)

Remote command:
CALCulate<Chn>:LIMit:DCIRcle:CLEar

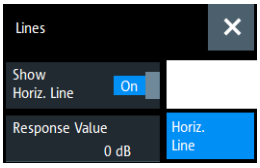
Draw Circle / Radius, Center X, Center Y
Defines the Display Circle – either by drawing it in the diagram area or by providing its radius and center.

Remote command:
CALCulate<Chn>:LIMit:DCIRcle:DATA

5.6.5 Horiz. Line tab

Shows or hides a horizontal line associated to the active trace in a cartesian diagram area. The line can be moved to particular trace points to retrieve the response values. It also shows which parts of a trace are above or below a definite response value.

- The controls on this tab are only active if the active trace is displayed in cartesian format.
- If another trace format is selected, the line (position) is deleted.



Show Horiz. Line
Displays or hides the horizontal line.

Remote command:
CALCulate<Chn>:DLINe:STATe

Response Value
Defines/shows the response value of the horizontal line.

Tip: Use the R&S ZNB's drag and drop functionality to move the horizontal line to a particular position. The response value appears in the numeric entry field.

Remote command:
CALCulate<Chn>:DLINe

5.7 Marker softtool

The "Marker" softtool allows you to position markers on a trace and to define their properties. Markers are also convenient tools for searching special points on traces and for scaling diagrams.

Access: Trace – [Marker]

Background information

Refer to the following sections:

- Section 4.2.1.3, "Markers", on page 96
- Section 3.3.7.6, "Set by marker", on page 68

5.7.1 Markers tab

Creates markers and configures their properties. Markers are available for all trace formats.

A first marker labeled "M1" is automatically created when the [Marker] hardkey is pressed. The "Mkr 1" ... "Mkr 10" and "Ref Mkr" softkeys enable the corresponding markers.

"Add Marker" adds a new marker.

Related information

Refer to the following sections:

- Section 4.2.1.3, "Markers", on page 96
- Section 3.3.5, "Handling diagrams, traces, and markers", on page 57



Mkr <i> Stimulus / Ref Mkr Stimulus
Gets/sets the stimulus value of the active marker.

Remote command:
`CALCulate<Chn>:MARKer<Mk>:X`
`CALCulate<Chn>:MARKer<Mk>:REFerence:X`

Mkr <i> Arb. Response / Ref Mkr Arb. Response
Gets/sets the response value (Y position) of an "Arbitrary" marker (see "Marker Mode" on page 507).

Remote command:
`CALCulate<Chn>:MARKer<Mk>:Y`
`CALCulate<Chn>:MARKer<Mk>:REFerence:Y`

On
Enables/disables the active marker.

Markers remember their "Marker Props" while disabled (see Section 5.7.2, "Marker Props tab", on page 506). The marker properties are definitely lost when the associated trace is deleted.

Remote command:
`CALCulate<Chn>:MARKer<Mk>[:STATe]`
`CALCulate<Chn>:MARKer<Mk>:REFerence[:STATe]`

All Off
Disables all markers of the active trace.

Markers remember their "Marker Props" while disabled (see Section 5.7.2, "Marker Props tab", on page 506). The marker properties are definitely lost when the associated trace is deleted.

Tip: To disable a single marker, drag it into vertical direction to release it from the trace and drop it onto the "Delete" icon.

Remote command:
`CALCulate<Chn>:MARKer<Mk>:AOFF`

Delta Mode
Enables/disables the "Delta Mode" for the active marker. At the same time, enables the Ref Mkr.

This function is inactive if the reference marker is the active marker.

Remote command:
`CALCulate<Chn>:MARKer<Mk>:DELTa[:STATe]`

Mkr 1 ... Mkr 10
Creates the markers numbered 1 to 10 and assigns them to the active trace. When a marker is created, a triangle labeled "M<i>" is positioned on the trace and the marker coordinates are displayed in the movable info field.

The stimulus position of an active marker can be entered in the "Mkr <i> Stimulus" entry field. The default position is the center of the sweep range. You can also drag and drop markers in a diagram to change their X position.

If the Marker Mode is "Arbitrary", also the response value (Y position) can be changed.

See also "Activating and moving markers" on page 97.

Remote command:
`CALCulate<Chn>:MARKer<Mk>[:STATe]`
`CALCulate<Chn>:MARKer<Mk>:Y`
`CALCulate<Chn>:MARKer[:STATe]:AREA`
`DISPlay[:WINDow<Wnd>]:MARKer:CATalog?`

Ref Mkr
Creates a reference marker and assigns it to the active trace. When a marker is created, a triangle labeled "R" is positioned on the trace and the marker coordinates are displayed in the info field.

The stimulus position of the active reference marker can be entered in the "Ref Marker Stimulus" entry field. The default position is the start of the sweep range or the position of the last active marker.

The reference marker defines the reference value for all markers that are in "Delta Mode".

Remote command:
`CALCulate<Chn>:MARKer<Mk>:REFerence[:STATe]`
`CALCulate<Chn>:MARKer<Mk>:REFerence:Y`

Add Marker
Adds a new marker to the active trace. Uses the next "free" marker number

Coupled Markers
Activates or deactivates Marker coupling. The label indicates the selected Coupling Type:

"Coupling Type"	Label
"All"	"Coupled Markers"
"Channel"	"Coupled Markers in <name of active channel>"
"Diagram"	"Coupled Markers in <name of active diagram>"

Coupling also works if Tracking is enabled for *Marker <m>* in **one** trace. The *Marker <m>* of the other traces then follow the movements of the tracked marker. The same holds true for the reference marker.

Remote command:
`CALCulate:MARKer:COUPled[:STATe]`

5.7.2 Marker Props tab

Modifies the properties of a marker created previously (see Section 5.7.1, "Markers tab", on page 503). The functions (except "Export Markers...") are unavailable if the active trace contains no markers.



Marker Name
Assigns a (new) name to the active marker. Marker names can contain letters, numbers, blanks and special characters.

Remote command:
`CALCulate<Chn>:MARKer<Mk>:NAME`
`CALCulate<Chn>:MARKer<Mk>:REFerence:NAME`

Marker Format
Defines the formatting of the active marker in the movable marker info field.

For background information on marker formats, see "Marker format" on page 99.

"Default" means that the marker is formatted according to the related trace's Dflt Marker Frmt.

In "Arbitrary" Marker Mode, if the transformation between trace format and marker format requires a concrete stimulus value, some result values in the marker info field can be unavailable. Those values are displayed as a sequence of dashes (------).

Remote command:
`CALCulate<Chn>:MARKer<Mk>:FORMat`
`CALCulate<Chn>:MARKer<Mk>:REFerence:FORMat`

Marker Style
Defines how the selected marker is displayed on the screen.

Remote command:
`CALCulate<Chn>:MARKer<Mk>:STYLe`

Discrete
Discrete mode means that a marker can be set to discrete sweep points only. If discrete mode is switched off, the marker can be positioned on any point of the trace, and its response values are obtained by interpolation.

Remote command:
`CALCulate<Chn>:MARKer<Mk>:MODE`
`CALCulate<Chn>:MARKer<Mk>:REFerence:MODE`

Marker Mode
Determines if and how the marker's position is adjusted and if and how it can be moved in the diagram area.

Normal: If Tracking is enabled, the marker's stimulus value is updated automatically with every sweep, otherwise it is constant. The marker position is adjusted to the corresponding response value, i.e. the marker is always positioned on the trace.

If in the current trace format the X axis represents the stimulus, the marker can be moved horizontally. At the same time, the marker's stimulus and response values are adjusted, i.e. its vertical position automatically follows the trace.

Fixed: freezes the marker at the position determined by the current stimulus and response value. Tracking is disabled. The stimulus and response values are stored with the marker; they are not adjusted to subsequent sweeps or trace format changes.

If in the current trace format the X axis represents the stimulus, the marker can be moved horizontally. Moving the marker adjusts the markers's stimulus value, but its response value remains fixed.

Arbitrary: freezes the marker at the position determined by the current stimulus and response value. Tracking is disabled. The marker stores the stimulus value and – in addition – its X and Y coordinates in the current Marker Format.

The marker can be moved freely inside the diagram, directly adjusting its X and Y coordinates. If in the current trace format the X axis represents the stimulus, the marker's stimulus value is adjusted accordingly. Otherwise the marker's stimulus value remains unchanged and is not shown in the Marker info field. Switching between trace formats resets the marker position to the response value at the marker's stimulus value, i.e. the marker snaps to the trace.

Remote command:
`CALCulate<Chn>:MARKer<Mk>:TYPE`
`CALCulate<Chn>:MARKer<Mk>:REFerence:TYPE`

Marker Info
Enables or disables the display of the dedicated marker info field at the marker symbol.

Remote command:
`CALCulate<Chn>:MARKer<Mk>:INFO`

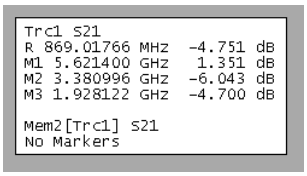
Ref Mkr -> Mkr
Places the reference marker to the position of the active marker. "Ref. Mkr -> Mkr" is not active if the active marker is a reference marker.

Remote command:
`CALCulate<Chn>:MARKer<Mk>:FUNCtion:SETRef`

Export Markers
Calls up a "Save As" dialog to store the current marker values to a marker file.

The analyzer uses a simple ASCII format to export marker values. By default, the marker file extension is `*.txt`. The file contains all traces in the active recall set together with their names and measured quantities. Below each trace, the file shows a list of all markers with their names, stimulus and response values.

The following example of a marker file describes a recall set with two traces, "Trc1" and its memory trace "Mem2[Trc1]". "Trc1" has a reference marker "R" and three normal markers "M1, M2, M3" assigned, the memory trace has no markers.



Remote command:
`MMEMory:STORe:MARKer`

Decimal Places...
Opens the "User Interface" tab of the "System Config" dialog, which allows to define the (maximum) number of decimal digits for different units. See "User Interface tab" on page 815.

5.7.3 Marker Search tab

Provides "Marker Search" functions that move the active marker to a (local) maximum or minimum of the active trace.

The search operation can be restricted to a configurable range of stimulus values ("Search Range..."). By default, the search range is equal to the entire sweep range.

If necessary, the active marker is enabled to indicate the search result.

Background information

Refer to "Basic marker search functions" on page 101.

5.7.3.1 Controls on the Marker Search tab



Max / Min
Sets the active marker to the absolute maximum or minimum in the search range, i.e. to the largest or smallest of all response values. If a complex trace format (e.g. a polar diagram) is active, the marker is set to the measurement point with the maximum or minimum magnitude.

"Max" and "Min" also overwrite the current "Search Mode" (--> "Search Min" and "Search Max") and the "Peak Type" for the peak search functions.

Remote command:

```
CALCulate<Chn>:MARKer<Mk>:FUNCtion:EXECute MINimum | MAXimum
CALCulate<Chn>:MARKer<Mk>:REFerence:FUNCtion:EXECute MINimum |
MAXimum
```

```
CALCulate<Chn>:MARKer<Mk>:FUNCtion:RESult?
CALCulate<Chn>:MARKer<Mk>:REFerence:FUNCtion:RESult?
```

Center = Marker
See "Center = Marker / Start = Marker / Stop = Marker / Span = Marker" on page 523.

Next Peak
Sets the active marker to the next local maximum or minimum in the search range, depending on the selected Peak Type.

Remote command:
```
CALCulate<Chn>:MARKer<Mk>:FUNCtion:EXECute NPEak
CALCulate<Chn>:MARKer<Mk>:REFerence:FUNCtion:EXECute NPEak
CALCulate<Chn>:MARKer<Mk>:FUNCtion:RESult?
CALCulate<Chn>:MARKer<Mk>:REFerence:FUNCtion:RESult?
```

Peak Left / Peak Right
Sets the active marker to the next local maximum or minimum to the left or right of the current marker position, depending on the selected Peak Type.

Remote command:
```
CALCulate<Chn>:MARKer<Mk>:FUNCtion:EXECute LPEak | RPEak
CALCulate<Chn>:MARKer<Mk>:REFerence:FUNCtion:EXECute LPEak |
RPEak
CALCulate<Chn>:MARKer<Mk>:FUNCtion:RESult?
CALCulate<Chn>:MARKer<Mk>:REFerence:FUNCtion:RESult?
```

Peak Type
Defines the peak type to be searched for using Next Peak and Peak Left / Peak Right:
- If "Max" is active, then the marker is set to the next maximum. The next maximum is the maximum with the largest response value that is below the current marker response value.
- If "Min" is active, then the marker is set to the next minimum. The next minimum is the minimum with the smallest response value that is above the current marker response value.
- If "Min or Max" is active, then the marker is set to the next minimum or maximum, whichever has the smallest distance from the current marker response value.

Remote command:
```
CALCulate<Chn>:MARKer<Mk>:FUNCtion:PEAK
```

Search Range...
Opens the Search Range dialog.

Tracking
Enables/disables tracking of the active marker for the current Search Mode, which causes the marker to be updated after each sweep (or after each sweep point in case of "Sweep Progress").

Among the available search modes, the tracking functionality only makes sense for:
- "Min" and "Max", where such an update typically causes the marker to change both its horizontal and vertical position and

- "Target Search", where typically only the horizontal position changes
- "Sweep Progress" for long duration sweeps

Define an adequate "Search Range" to restrict the search to the adequate frequency or power interval (see Section 5.7.3.2, "Search Range dialog", on page 511).

Note: Tracking for bandfilter search can be activated separately, see "Tracking" on page 522.

Remote command:

```
CALCulate<Chn>:MARKer<Mk>:SEARch:TRACking
CALCulate<Chn>:MARKer<Mk>:REFerence:SEARch:TRACking
```

Marker Config...
Opens the Marker Config dialog.

5.7.3.2 Search Range dialog

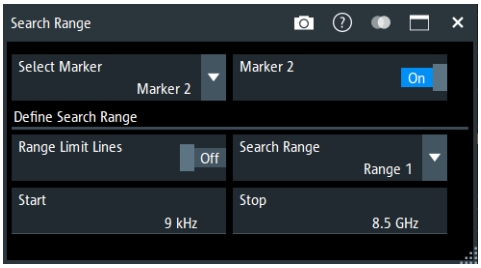
The "Search Range" dialog confines the "Marker Search" and "Target Search" for the selected marker to a subrange of the sweep. The search range is a continuous interval of the sweep variable.

Access:

- Trace – [Marker] > "Marker Search" > "Search Range..."
- Trace – [Marker] > "Target Search" > "Search Range..."

If Tracking is active, the assigned "Search Range" applies to all sweeps and can be used to achieve uniqueness in "Min", "Max" or "Target Search".

See also Section 5.5.5, "Trace Statistics tab", on page 465.



It is possible to define search ranges for each recall set and assign them to the corresponding markers and the reference marker.

Select Marker
Selects the reference marker or one of the numbered markers that can be assigned to the trace. If a numbered marker does not exist, it is created when "On" is enabled. A created marker is displayed in the center of the search range.

Search Range
Selects the "Search Range" to be assigned to the selected marker. "Full Span" means that the "Search Range" is equal to the sweep range. Besides, it is possible to store up to 10 customized search ranges.

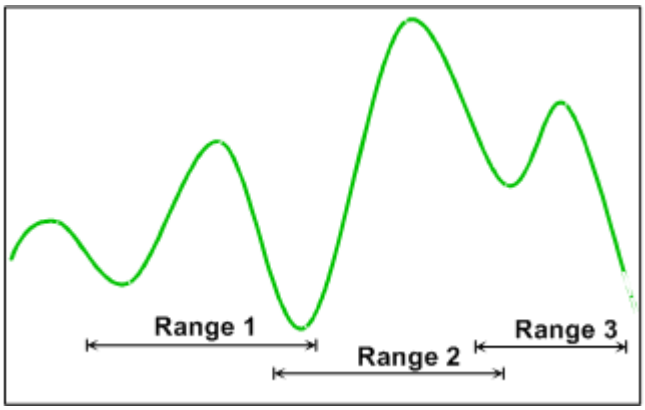
The "Search Range"s are bordered by the "Start" and "Stop" values. "Start" must be smaller than "Stop", otherwise the second value is automatically adjusted.

"Search Range" properties

The 10 search ranges are valid for the entire recall set. Each of them can be assigned to any marker in the recall set, irrespective of the trace and channel that the marker belongs to.

The default search range of any new marker is "Full Span". The analyzer provides the greatest flexibility in defining search ranges. In particular, two search ranges can overlap or even be identical. The search is confined to the part of the search range that belongs to the sweep range.

The following example shows how "Search Range"s can be used to search a trace for several local maxima.



Note: The marker Search Ranges are identical to the evaluation ranges for trace statistics. For more information, see Section 5.5.5.2, "Evaluation Range dialog", on page 470.

Remote command:

```
CALCulate<Chn>:MARKer<Mk>:FUNCtion:DOMain:USER[:RANGe]
CALCulate<Chn>:MARKer<Mk>:REFerence:FUNCtion:DOMain:USER[:RANGe]
CALCulate<Chn>:MARKer<Mk>:FUNCtion:DOMain:USER:STARt
CALCulate<Chn>:MARKer<Mk>:REFerence:FUNCtion:DOMain:USER:STARt
CALCulate<Chn>:MARKer<Mk>:FUNCtion:DOMain:USER:STOP
CALCulate<Chn>:MARKer<Mk>:REFerence:FUNCtion:DOMain:USER:STOP
```

Range Limit Lines On

Displays or hides the range limit lines in the diagram area. Range limit lines are two vertical lines at the Start and Stop values of the active search range ("Range 1" to "Range 10").

The line color can be modified by a user-defined color scheme (Element "Vertical Range Lines").

Remote command:

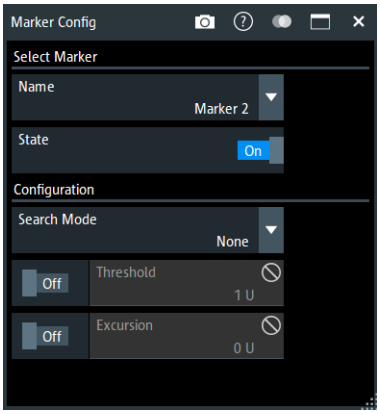
```
CALCulate<Chn>:MARKer<Mk>:FUNCtion:DOMain:USER:SHOW
CALCulate<Chn>:MARKer<Mk>:REFerence:FUNCtion:DOMain:USER:SHOW
```

5.7.3.3 Marker Config dialog

The "Marker Config" dialog allows you to configure the peak searches for the active trace and all its markers.

Access:

- Trace – [Marker] > "Marker Search" > "Marker Config..."
- Trace – [Marker] > "Multiple Peak" > "Marker Config..."



Select Marker
Allows you to select the related marker and to activate or deactivate it.

Remote command:
`CALCulate<Chn>:MARKer<Mk>[:STATe]`

Search Config
Allows you to select the Target Search Mode of the selected marker.

Remote command:
`CALCulate<Chn>:MARKer<Mk>:FUNCtion:EXECute`
`CALCulate<Chn>:MARKer<Mk>:REFerence:FUNCtion:EXECute`

Threshold Settings
Defines a threshold for (single) peak searches and activates it.

If "Active", only maxima above and minima below the configured "Threshold" are considered.

The unit is derived from the active trace format and cannot be modified. Changing the trace format resets the "Threshold" to a format-specific default value.

Remote command:
`CALCulate<Chn>:MARKer<Mk>:THReshold`
`CALCulate<Chn>:MARKer<Mk>:REFerence:THReshold`
`CALCulate<Chn>:MARKer<Mk>:THReshold:STATe`
`CALCulate<Chn>:MARKer<Mk>:REFerence:THReshold:STATe`

Excursion Settings
Defines a minimum excursion for peak searches and activates it.

If "Active", only peaks with an excursion above the configured "Excursion" value are considered. By definition, the excursion of a peak is the smaller of the absolute differences in measured values from the adjoining peaks of opposite polarity.

The unit is derived from the active trace format and cannot be modified. Changing the trace format resets the "Excursion" to a format-specific default value.

Remote command:

```
CALCulate<Chn>:MARKer<Mk>:EXCursion
CALCulate<Chn>:MARKer<Mk>:REFerence:EXCursion
CALCulate<Chn>:MARKer<Mk>:EXCursion:STATe
CALCulate<Chn>:MARKer<Mk>:REFerence:EXCursion:STATe
```

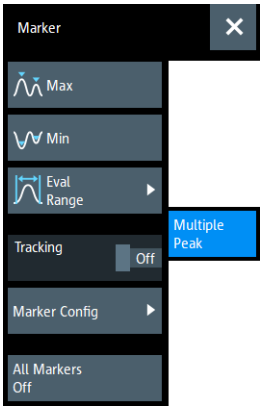
5.7.4 Multiple Peak tab

"Multiple Peak" search allows you to find multiple local minima/maxima at once.

Background information

Refer to "Basic marker search functions" on page 101.

5.7.4.1 Controls on the Multiple Peak tab



Max / Min

Sets up to 10 markers to the highest maxima or lowest minima in the configured Eval Range. If a complex trace format is active (e.g. a polar diagram), the markers are set to the measurement points with the maximum or minimum magnitude.

The required markers are created/deleted as needed.

Remote command:

```
CALCulate<Chn>:MARKer<Mk>:FUNCtion:EXECute MMAXimum | MMINimum
CALCulate<Chn>:MARKer<Mk>:REFerence:FUNCtion:EXECute MMAXimum |
MMINimum
CALCulate<Chn>:MARKer<Mk>:FUNCtion:RESult?
CALCulate<Chn>:MARKer<Mk>:REFerence:FUNCtion:RESult?
```

Eval Range
Opens the Evaluation Range dialog that allows you to set the domain for the multiple peak search. A modified domain takes effect the next time Max / Min is used.

Tracking
Enables or disables tracking for "Multiple Peak" search. If enabled, a new multiple peak search is performed for each sweep (creating/deleting markers as needed).

Define an Eval Range to restrict the search to the adequate frequency or power interval.

Remote command:
`CALCulate<Chn>:MARKer<Mk>:SEARch:TRACking`

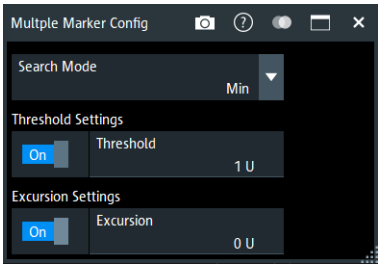
Marker Config...
Opens the Multiple Marker Config dialog.

All Markers Off
See "All Off" on page 504.

5.7.4.2 Multiple Marker Config dialog

The "Multiple Marker Config" dialog allows you to configure the multiple peak searches for the active trace.

Access: Trace – [Marker] > "Multiple Peak" > "Marker Config..."



Search Config
Same as selecting Max / Min.

Remote command:
`CALCulate<Chn>:MARKer<Mk>:FUNCtion:EXECute MMAXimum | MMINimum`
`CALCulate<Chn>:MARKer<Mk>:REFerence:FUNCtion:EXECute MMAXimum | MMINimum`

Threshold Settings
Defines a threshold for multiple peak searches and activates it.

If "Active", only maxima above and minima below the configured "Threshold" are considered.

The unit is derived from the active trace format and cannot be modified. Changing the trace format resets the "Threshold" to a format-specific default value.

Remote command:
CALCulate<Chn>:MARKer:MPEak:THReshold
CALCulate<Chn>:MARKer:MPEak:THReshold:STATe

Excursion Settings
Defines a minimum excursion for multiple peak searches and activates it.

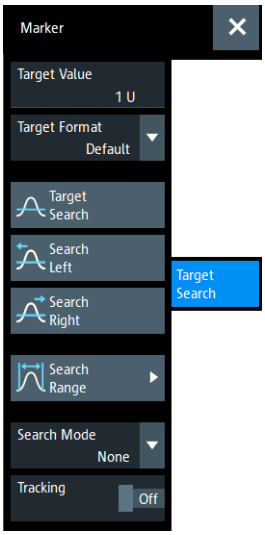
If "Active", only peaks with an excursion above the configured "Excursion" value are considered. By definition, the excursion of a peak is the smaller of the absolute differences in measured values from the adjoining peaks of opposite polarity.

The unit is derived from the active trace format and cannot be modified. Changing the trace format resets the "Excursion" to a format-specific default value.

Remote command:
CALCulate<Chn>:MARKer:MPEak:EXCursion
CALCulate<Chn>:MARKer:MPEak:EXCursion:STATe

5.7.5 Target Search tab

The "Target Search" functions use markers to locate trace points with a specific response value ("Target Value"). The functions are unavailable if the active trace contains no markers (e.g. after "All Markers Off").



Some of the "Target Search" functions are equal to other marker search functions. Refer to the following sections:

- Section 5.7.3.2, "Search Range dialog", on page 511
- "Tracking" on page 510

Target Value
Specifies the target value for the search.

The VNA software allows you to specify the target value in different formats (see Target Format below). For example, you can search for a particular phase value in a Smith chart.

Remote command:
CALCulate<Chn>:MARKer<Mk>:TARGet
CALCulate<Chn>:MARKer<Mk>:REFerence:TARGet

Target Format
Selects the format that is used to specify the Target Value.

The selected target format applies to the current marker only: each marker can have a different target format. The table below gives an overview on how a complex target value z = x + jy is converted.

Target Format	Description	Formula
"Lin Mag"	Magnitude of z, unconverted.	$\|z\| = \sqrt{(x^2 + y^2)}$
"dB Mag"	Magnitude of z [dB]	Mag(z) = 20 log|z| dB
"Phase"	Phase of z [°]	φ (z) = arctan (y/x)
"Phase unwrap"	Unwrapped phase of z comprising the complete number of 360° phase rotations [°]	Φ(z) = φ (z) + 2k·360°
"Real"	Real part of z	Re(z) = x
"Imag"	Imaginary part of z	Im(z) = y
"SWR"	(Voltage) Standing Wave Ratio	SWR = (1 + |z|) / (1 – |z|)
"Default"	Identical to trace format. **Note**: the Smith and Polar traces use "Lin Mag" as the default format for target value.	-

Remote command:
CALCulate<Chn>:MARKer<Mk>:SEARch:FORMat
CALCulate<Chn>:MARKer<Mk>:REFerence:SEARch:FORMat

Target Search
Activates the search and sets the active marker to the defined target value. If the target value occurs at several stimulus values, the marker is placed to the search result with the smallest stimulus value. The other measurement points with the same target value can be located using the "Search Right" function.

If the target is not found (e.g. because the active trace does not contain the target value), then the active marker is not moved away from its original position.

Remote command:
CALCulate<Chn>:MARKer<Mk>:FUNCtion:EXECute TARGet
CALCulate<Chn>:MARKer<Mk>:REFerence:FUNCtion:EXECute TARGet
CALCulate<Chn>:MARKer<Mk>:FUNCtion:RESult?
CALCulate<Chn>:MARKer<Mk>:REFerence:FUNCtion:RESult?

Search Left/Search Right
Searches the Target Value to the left/right of the active marker's stimulus value within the current search range (see Section 5.7.3.2, "Search Range dialog", on page 511).

If the search is successful, the active marker is moved to the next smaller/larger stimulus value with this target value. Use "Search Left"/"Search Right" repeatedly to locate the other ones.

Remote command:

```
CALCulate<Chn>:MARKer<Mk>:FUNCtion:EXECute LTARget
CALCulate<Chn>:MARKer<Mk>:REFerence:FUNCtion:EXECute LTARget
CALCulate<Chn>:MARKer<Mk>:FUNCtion:RESult?
CALCulate<Chn>:MARKer<Mk>:REFerence:FUNCtion:RESult?
```

Search Mode
Displays and sets the current marker search mode.

Select one of the predefined max, min, peak, or target searches or select "Sweep Progress" to track the position of the sweep cursor.

Remote command:

```
CALCulate<Chn>:MARKer<Mk>:FUNCtion:EXECute
CALCulate<Chn>:MARKer<Mk>:REFerence:FUNCtion:EXECute
```

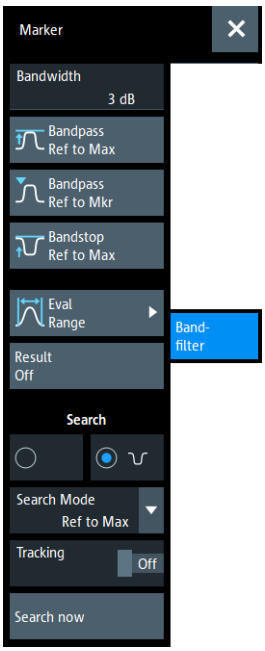
5.7.6 Bandfilter tab

"Bandfilter" search allows you to search for trace segments with a bandpass or bandstop shape, and determine characteristic filter parameters.

Background information
Refer to "Bandfilter search" on page 102.

Bandfilter for arbitrary scalar traces

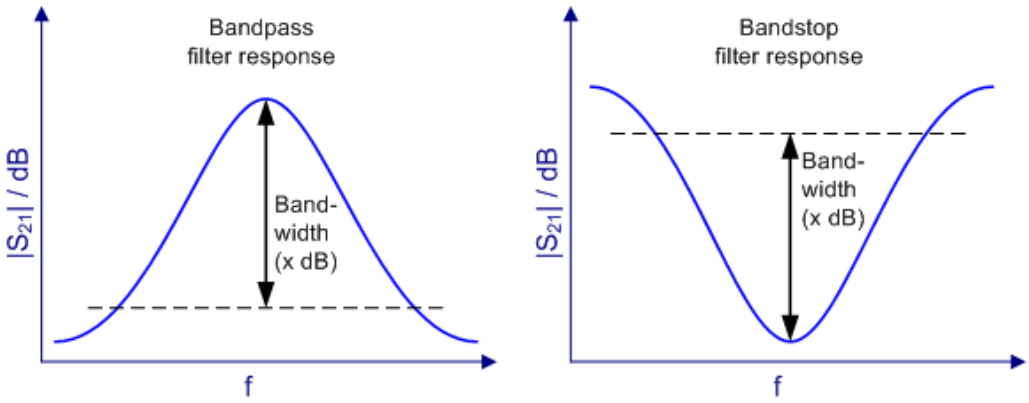
"Bandfilter" search can be used for a broad range of measured quantities (see Section 5.2, "Meas softtool", on page 291). To obtain real filter parameters, the trace format must be "dB Mag", the measured quantity must be a transmission S-parameter and a frequency sweep must be performed. However, for other formats, measured quantities or sweep types, the "Bandfilter" functions can still be useful to analyze general trace properties.



The "Eval Range..." button opens the "Evaluation Range" dialog that allows you to narrow the "Bandfilter" search to a particular stimulus range. See Evaluation Range dialog.

Bandwidth
Specifies the minimum excursion of the bandpass and bandstop peaks.

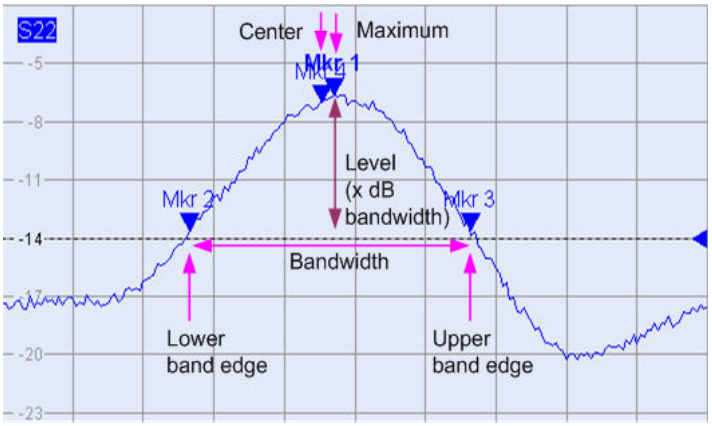
- A bandpass peak must fall off on both sides by the specified <Bandwidth> value to be considered a valid peak.
- A bandstop peak must be <Bandwidth> below the maximum level in the search range (bandpass value) to be considered a valid peak.



Remote command:
`CALCulate<Chn>:MARKer<Mk>:BWIDth`

Bandpass Ref to Max
Activates the search for a bandpass region on the active trace and activates Tracking. The located bandpass region is the tallest peak in the search range with a minimum excursion as specified by the "Bandwidth" parameter.

If "Bandpass Ref to Max" is selected, the analyzer uses (or creates) the four markers "M1" to "M4" to locate the **bandpass region**.



- "M1" indicates the maximum of the peak ("Max").
- "M2" indicates the point on the left edge of the peak where the trace value is equal to the maximum minus the bandwidth factor ("Lower Edge").
- "M3" indicates the point on the right edge of the peak where the trace value is equal to the maximum minus the bandwidth factor ("Upper Edge").
- "M4" indicates the center of the peak. Depending on a system setting, the center is either calculated as the geometric or the arithmetic mean of the "Lower Edge" and "Upper Edge" frequencies (see "Geometric Calculation of Bandfilter Center" on page 818).

The search results are displayed in the movable "Bandfilter" info field.

Remote command:

```
CALCulate<Chn>:MARKer<Mk>:FUNCtion:BWIDth:MODE BPASs
CALCulate<Chn>:MARKer<Mk>:FUNCtion:EXECute BFILter
CALCulate<Chn>:MARKer:SEARch:BFILter:RESult[:STATe]:AREA
```

Bandpass Ref to Mkr

Activates the search for a bandpass region on the active trace and activates Tracking, starting at the position of the active marker. A bandpass region is the closest peak in the evaluation range that has a minimum excursion as specified by the "Bandwidth" parameter.

In contrast to a "Bandpass Ref to Max", the "Bandpass Ref to Mkr" does not change the position of the active markers. The search results are displayed in the movable "Bandfilter" info field.

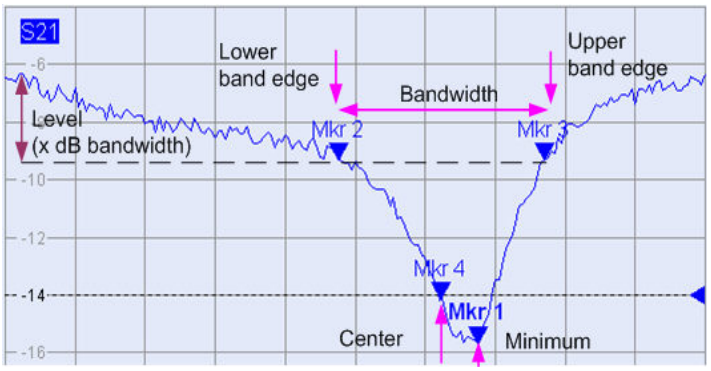
Remote command:

```
CALCulate<Chn>:MARKer<Mk>:FUNCtion:BWIDth:MODE BPRMarker
CALCulate<Chn>:MARKer<Mk>:FUNCtion:EXECute BFILter
```

Bandstop Ref to Max

Activates the search for a bandstop region on the active trace and activates Tracking. A bandstop region is the lowest peak (local minimum) in the search range, whose level is at least <Bandwidth> below the maximum (passband value).

If "Bandstop Ref to Max" is selected, the analyzer uses (or creates) the four markers "M1" to "M4" to locate the **bandstop region**.



- "M1" indicates the minimum of the peak ("Min").
- "M2" indicates the "Lower Edge" of the bandstop, i.e. the point on the left edge of the peak where the trace value is equal to the maximum in the search range (pass-band value) minus the specified Bandwidth.
- "M3" indicates the "Upper Edge" of the bandstop, i.e. the point on the right edge of the peak where the trace value is equal to the maximum in the search range (pass-band value) minus the specified "Bandwidth".
- "M4" indicates the center of the peak. Depending on a system setting, the center is either calculated as the geometric or the arithmetic mean of the "Lower Edge" and "Upper Edge" positions (see "Geometric Calculation of Bandfilter Center" on page 818).

The search results are displayed in the movable "Bandfilter" info field.

Remote command:

```
CALCulate<Chn>:MARKer<Mk>:FUNCtion:BWIDth:MODE BSTop
CALCulate<Chn>:MARKer<Mk>:FUNCtion:EXECute BFILter
```

Result Off

Hides the movable info field with the results of a bandpass or a bandstop search and disables Tracking. The info field is displayed again (and tracking re-enabled) when a new "Bandfilter" search is performed.

Remote command:

```
CALCulate<Chn>:MARKer:SEARch:BFILter:RESult[:STATe]
```

Search (Bandpass or Bandstop) / Search Mode

Enables a bandpass or bandstop search (left/right icon) for an arbitrary search mode. The search modes have the following effect:

- "Ref to Max": The bandpass (bandstop) is the tallest (lowest peak) in the search range. For a detailed description, refer to "Bandpass Ref to Max" on page 519 and "Bandstop Ref to Max" on page 520.
- "Ref to Marker": The bandpass (bandstop) is the tallest (lowest) peak in the search range. The response value for the lower and upper band edges is calculated as the response value at the active marker position plus (minus) the Bandwidth. To be valid, the peak must be above (below) the response value for the band edges.

- "Absolute Level:" The bandpass (bandstop) is the tallest (lowest) peak in the search range. To be valid, the peak must be above (below) -"Bandwidth". The Lower Band Edge and Upper Band Edge values are given by the frequencies where the trace is equal to -"Bandwidth".
- "None": "Bandfilter" search switched off, result off.

Remote command:
`CALCulate<Chn>:MARKer<Mk>:FUNCtion:BWIDth:MODE`

Tracking
Causes the active "Bandfilter" search to be repeated after each sweep: When tracking mode is active, the markers typically change their horizontal and vertical positions as the measurement proceeds.

Tracking mode properties

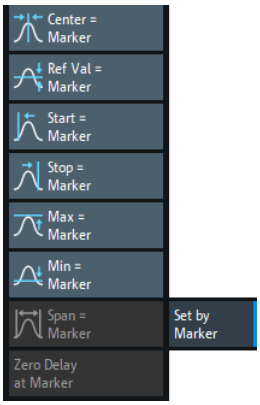
Tracking modes are available for all search modes. The tracking modes for minimum/ maximum/peak search and target search are coupled; tracking for "Bandfilter" search can be activated separately. Tracking is activated automatically when one of the "Band-filter" search modes is selected.

Remote command:
"Bandfilter" tracking and marker/target search tracking are controlled with the same command:
`CALCulate<Chn>:MARKer<Mk>:SEARch:TRACking`

5.7.7 Set by Marker tab

The "Set by Marker" functions use the active marker to define the sweep range, scale the diagram and introduce an electrical length offset. The functions are unavailable if the active trace contains no markers (e.g. after "All Markers Off").

Examples
Refer to Section 3.3.7.6, "Set by marker", on page 68.



Center = Marker / Start = Marker / Stop = Marker / Span = Marker
The following functions use the stimulus value of the active marker to define the sweep range.

- "Center = Marker" sets the center of the sweep range equal to the stimulus value of the active marker, leaving the span unchanged. The active marker appears in the center of the diagram.
- "Start = Marker" sets the beginning (start) of the sweep range equal to the stimulus value of the active marker, leaving the end (stop) value unchanged. The active marker appears at the left edge of the diagram.
- "Stop = Marker" sets the end (stop) of the sweep range equal to the stimulus value of the active marker, leaving the beginning (start) unchanged. The active marker appears at the right edge of the diagram.
- "Span = Marker" is only available for frequency sweeps (linear or logarithmic). It is enabled if the active marker is in Delta Mode. "Span = Marker" adjusts the sweep span to the range between the active delta marker and the reference marker.

Remote command:
```
CALCulate<Chn>:MARKer<Mk>:FUNCtion:CENTer
CALCulate<Chn>:MARKer<Mk>:REFerence:FUNCtion:CENTer
CALCulate<Chn>:MARKer<Mk>:FUNCtion:STARt
CALCulate<Chn>:MARKer<Mk>:REFerence:FUNCtion:STARt
CALCulate<Chn>:MARKer<Mk>:FUNCtion:STOP
CALCulate<Chn>:MARKer<Mk>:REFerence:FUNCtion:STOP
CALCulate<Chn>:MARKer<Mk>:FUNCtion:SPAN
```

Ref Val = Marker / Max = Marker / Min = Marker
The following functions use the response value of the active marker to scale the y-axis of the diagram:

- "Ref Val = Marker" sets the reference value equal to the response value of the active marker, leaving the values of the vertical divisions ("Scale / Div") unchanged.
- "Max = Marker" sets the upper edge of the diagram equal to the response value of the active marker, leaving the values of the vertical divisions ("Scale / Div") unchanged.
- "Min = Marker" sets the lower edge of the diagram equal to the response value of the active marker, leaving the values of the vertical divisions ("Scale / Div") unchanged.

Remote command:
```
CALCulate<Chn>:MARKer<Mk>:FUNCtion:REFVal
CALCulate<Chn>:MARKer<Mk>:REFerence:FUNCtion:REFVal
CALCulate<Chn>:MARKer<Mk>:FUNCtion:MAX
CALCulate<Chn>:MARKer<Mk>:REFerence:FUNCtion:MAX
CALCulate<Chn>:MARKer<Mk>:FUNCtion:MIN
CALCulate<Chn>:MARKer<Mk>:REFerence:FUNCtion:MIN
```

Zero Delay at Marker
This function is available for Delay traces only. It shifts the trace in vertical direction so that the delay at the marker position becomes zero. Mathematically, it modifies the measurement results by subtracting the delay at the current marker position.

The delay represents the propagation time of the wave. Hence "Zero Delay at Marker" performs an electrical length compensation, by adding or subtracting a simulated loss-less transmission line of variable length to or from the test port. This shift of the reference plane must be carried out on the "Delay" trace, but has an impact on all trace formats.

A standard application of "Zero Delay at Marker" is correction of the constant delay caused by the interconnecting cables between the analyzer test ports and the DUT (line stretch).

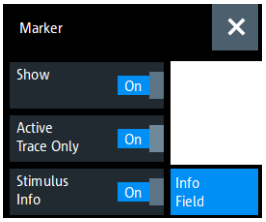
Note: "Zero Delay at Marker" modifies the "Offset" parameters and therefore influences the entire channel.

Remote command:
`CALCulate<Chn>:MARKer<Mk>:FUNCtion:ZERodelay`
`CALCulate<Chn>:MARKer<Mk>:REFerence:FUNCtion:ZERodelay`

5.7.8 Info Field tab

Displays or hides the marker info field and selects it contents. The functions are self-explanatory.

Background information
Refer to "Marker info field" on page 98



Show
If enabled, the marker info field is displayed.

Remote command:
`DISPlay[:WINDow<Wnd>]:ANNotation:MARKer:INFO[:STATe]`

Active Trace Only
If enabled, the marker info field displays only marker information for the active trace.

Remote command:
`DISPlay:ANNotation:MARKer:TRACe:INFO[:STATe]`

Stimulus Info
If enabled, the stimulus information (the frequencies of the markers) is included in the marker info field.

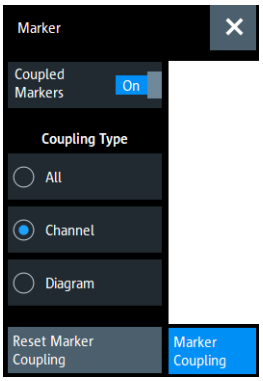
Remote command:
`DISPlay:ANNotation:MARKer:STIMulus[:STATe]`

5.7.9 Marker Coupling tab

Allows you to set up and control marker coupling.

Background information

Refer to "Marker coupling" on page 101.



Coupled Markers

Activates or deactivates Marker coupling. The label indicates the selected Coupling Type:

"Coupling Type"	Label
"All"	"Coupled Markers"
"Channel"	"Coupled Markers in <name of active channel>"
"Diagram"	"Coupled Markers in <name of active diagram>"

Coupling also works if Tracking is enabled for *Marker <m>* in **one** trace. The *Marker <m>* of the other traces then follow the movements of the tracked marker. The same holds true for the reference marker.

Remote command:

`CALCulate:MARKer:COUPled[:STATe]`

Coupling Type

Determines the set of traces whose markers are coupled to the markers of the active trace when marker coupling is enabled.

- **"All"**: marker coupling applies to all traces in the active recall set that have the same stimulus variable as the active trace
- **"Channel"**: marker coupling applies to all traces in the active channel
- **"Diagram"**: marker coupling applies to all traces in the active diagram that have the same stimulus variable as the active trace

The selected "Coupling Type" applies to all marker coupling relations in the current recall set. Changing the "Coupling Type" automatically disables marker coupling throughout the recall set.

Remote command:
`CALCulate:MARKer:COUPled:TYPE`

Reset Marker Coupling
Convenience function for disabling marker coupling and setting the Coupling Type to "All" (default).

Remote command:
`CALCulate:MARKer:COUPled[:STATe] OFF`
`CALCulate:MARKer:COUPled:TYPE ALL`

5.8 Stimulus softtool

On the "Stimulus" softtool, you can access to the stimulus parameters of the active channel. If the active trace is represented in Time Domain, it also allows you to configure the "observation interval".

Access: Stimulus – [Start] | [Stop] | [Center] | [Span] hardkey

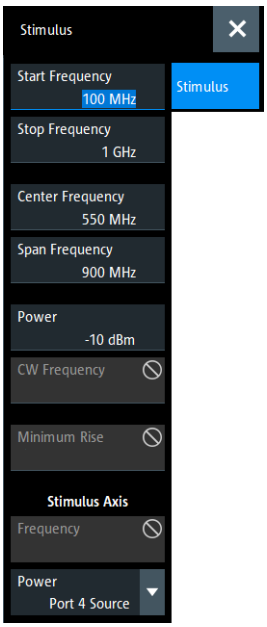
- While in Cartesian diagrams the x-axis represents the stimulus values, in polar and Smith diagrams this direct relation is lost. In any case, a marker can be used to display the stimulus value of a given trace point.
  For "Time Domain" traces, points in the "observation interval" are interpreted as stimulus values.
- All stimulus settings except the "Time Domain" settings are channel settings. "Time Domain" applies to the active (time domain) trace only.

Background information
Refer to the following sections:
- Section 4.1.3, "Traces, channels and diagrams", on page 80
- Section 3.3.7.2, "Setting the sweep range", on page 66
- Section 3.3.7.6, "Set by marker", on page 68

5.8.1 Stimulus tab

Defines the sweep range in the current channel, depending on the sweep type.

Related Settings
Refer to the following sections:
- Section 4.1.4.4, "Stimulus and sweep types", on page 87
- Section 5.10.2, "Sweep Type tab", on page 539

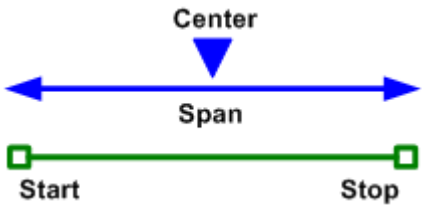


The following "Stimulus" settings are also available on the "Power" tab:

- Power
- Start Power / Stop Power

Start Frequency / Stop Frequency / Center Frequency / Span Frequency
Defines the sweep range for non-segmented frequency sweeps.

For a Lin Freq sweep, setting "Start Frequency" and "Stop Frequency" or "Center Frequency" and "Span Frequency" are alternatives.



For a Log Freq only "Start Frequency" and "Stop Frequency" can be set.

Note: For segmented frequency sweeps the start and stop frequencies and the number of sweep points are defined per segment. See Section 5.10.2.2, "Define Segments dialog", on page 545.

Remote command:
`[SENSe<Ch>:]FREQuency:STARt`
`[SENSe<Ch>:]FREQuency:STOP`
`[SENSe<Ch>:]FREQuency:CENTer`
`[SENSe<Ch>:]FREQuency:SPAN`
`SYSTem:FREQuency?` (query frequency range of the network analyzer)

Number of Points
Sets the total number of measurement points for CW Mode sweeps.

This value can also be set on the "Sweep Params" tab of the "Sweep"softtool (see "Number of Points" on page 537).

Remote command:
`[SENSe<Ch>:]SWEep:POINts`

Stop Time
Defines the sweep range for Time sweeps.

Remote command:
`[SENSe<Ch>:]SWEep:TIME`

CW Frequency
Sets the fixed frequency for Power, CW Mode, and Time sweeps.

The "CW Frequency" is also used as the channel base frequency for frequency-converting measurements; see Section 4.7.3, "Frequency conversion measurements", on page 233.

Remote command:
`[SENSe<Ch>:]FREQuency:FIXed`
`[SENSe<Ch>:]FREQuency[:CW]`
`SOURce<Ch>:FREQuency<PhyPt>:FIXed`
`SOURce<Ch>:FREQuency<PhyPt>[:CW]`

Stimulus Axis – Frequency / Power
Selects the channel base frequency (power) or the frequency (power) of a particular port as the stimulus axis in all diagrams of the active recall set. The selected stimulus axis is displayed in the channel line. Select the axis according to your port configuration and measurement results.

Example:

Port 1 provides the RF input signal for a mixer; the IF output signal is measured at Port 2. The port frequency of Port 1 is equal to the channel base frequency f_b, the port frequency of Port 2 is set to the IF frequency $f_b + f_{LO}$. To view the received wave b_2 over the entire IF frequency range, select port 2 as a frequency stimulus axis.

The lists include external generator and power meter ports. The "Frequency" list is disabled while the frequency conversion is switched off.

Remote command:
`[SENSe<Ch>:]SWEep:AXIS:FREQuency`
`[SENSe<Ch>:]SWEep:AXIS:POWer`

5.8.2 Power tab

The "Power" tab allows you to configure the signal power. It is identical to the "Power Bw Avg" > "Power" tab; see Section 5.9.1, "Source Power tab", on page 531.

5.8.3 Time Domain tab

If the active trace is a time domain trace, these settings define its stimulus axis.

Time domain analysis requires option R&S ZNB-K2 / R&S ZNBT-K2. If this option is not installed, the "Time Domain" tab is hidden.

Related information

Refer to the following sections:

- Section 4.7.1, "Time domain analysis", on page 218
- Section 5.2.5.1, "Time Domain tab", on page 356

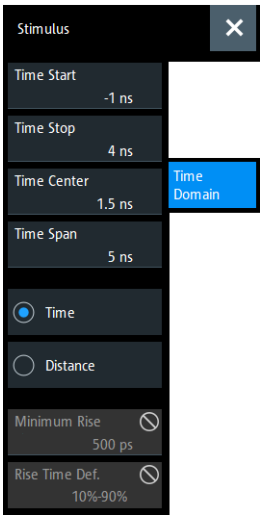


Figure 5-26: Stimulus > Time Domain X-Axis softtool tab

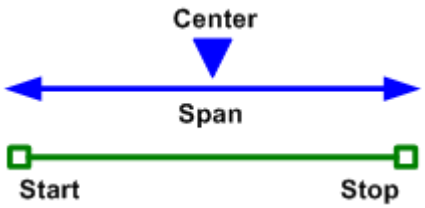
left = Time representation
right = Distance representation

Time Start / Time Stop / Time Center / Time Span

Defines the display range for the time domain trace in time representation (see "Time / Distance" on page 530).

- "Time Start" is the lowest displayed time and corresponds to the left edge of the Cartesian diagram.
- "Time Stop" is the highest displayed time and corresponds to the right edge of the Cartesian diagram.
- "Time Center" corresponds to the center of the Cartesian diagram, i.e. ("Time Start" + "Time Stop")/2.
- "Time Span" corresponds to the diagram width, i.e. ("Time Stop" – "Time Start").

"Time Start" and "Time Stop" or "Time Center" and "Time Span" are alternative settings.



Remote command:
CALCulate<Chn>:TRANsform:TIME:STARt
CALCulate<Chn>:TRANsform:TIME:STOP
CALCulate<Chn>:TRANsform:TIME:CENTer
CALCulate<Chn>:TRANsform:TIME:SPAN

Distance Start / Distance Stop / Distance Center / Distance Span
Defines the display range for the time domain trace in distance representation (see "Time / Distance" on page 530).

"Distance Start" and "Distance Stop" or "Distance Center" and "Distance Span" are alternative settings.

Remote command:
n.a.
See Time Start / Time Stop / Time Center / Time Span.

Time / Distance
"Time" and "Distance" switch over between the x-axis scaling in time units or distance units.

The interpretation of time and distance depends on the measurement type. For reflection measurements (S-parameters S_{ii} or ratios with equal port indices), the time axis represents the propagation time of a signal from the source to the DUT and back. For transmission measurement, it represents the propagation time from the source through the device to the receiver.

The distance between the source and the DUT is calculated from the propagation time, the velocity of light in the vacuum, and the velocity factor of the receiving port:

- *Distance = <Time> * c_0 * <Velocity Factor>* for transmission measurements
- *Distance = 1/2 * <Time> * c_0 * <Velocity Factor>* for reflection measurements. The factor 1/2 accounts for the return trip from the DUT to the receiver.

The velocity factor of the receiving port can be defined using Channel – [Offset Embed] > "Offset" (see Section 5.13.2.1, "Controls on the Offset tab", on page 738).

Remote command:
CALCulate<Chn>:TRANsform:TIME:XAXis

Rise Time/Rise Time Def.
Allows you to get and set the effective rise time for the configured time domain stimulus.

This parameter is only available for low pass transformation types. It is inversely related to the stop frequency.

Remote command:
[SENSe<Ch>:]HARMonic:RTIMe:THReshold
[SENSe<Ch>:]HARMonic:RTIMe:DATA

5.9 Power Bw Avg softtool

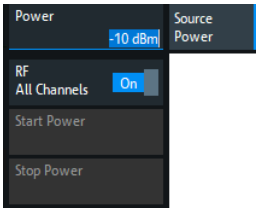
The "Power Bw Avg" softtool allows you to configure the signal power, to set up the IF signal processing, and to configure the averaging logic.

Access: Channel – [Pwr Bw Avg] hardkey

5.9.1 Source Power tab

The "Source Power" tab provides settings related to transmit power. It is also displayed on the "Power Bw Avg" softtool.

5.9.1.1 Controls on the Source Power tab



Power
Determines the output power at the test ports for the sweep types "Lin Freq", "Log Freq", "CW Mode", and "Time". Also determines the default output power for "Segmented" sweeps, where the output power can be set per segment.

The setting has no effect for Power sweeps, where the source power is varied over a continuous range.

Remote command:
SOURce<Ch>:POWer<PhyPt>[:LEVel][:IMMediate][:AMPLitude]

RF Off All Channels
"RF Off All Channels" switches the power sources for all channels off (if enabled) or on. Switching off the RF power helps to prevent overheating of a connected DUT while no measurement results are taken.

Tip: Switching off the internal RF sources while an external generator is used can improve the measurement accuracy. "RF Off All Channels" also deactivates external generators, so you have to use the settings in the "Arb Frequency" tab of the Port Settings dialog (with option R&S ZNB-K4).

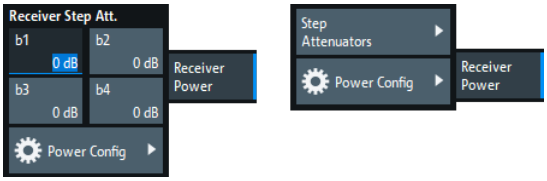
Remote command:
OUTPut<Ch>[:STATe]

Start Power / Stop Power
Defines the sweep range for Power sweeps.

Remote command:
SOURce<Ch>:POWer<PhyPt>:STARt
SOURce<Ch>:POWer<PhyPt>:STOP

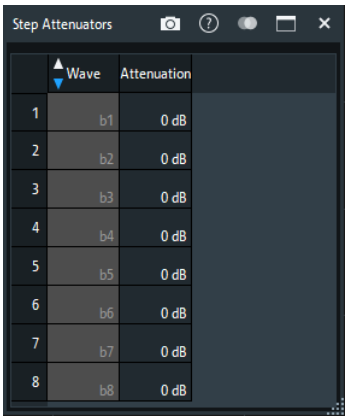
5.9.2 Receiver Power tab

This tab is only visible, if the R&S ZNB is equipped with Receiver step attenuators.



left = R&S ZNB without connected switch matrices
right = R&S ZNB with connected switch matrices

The "Step Attenuators..." button opens the following dialog:

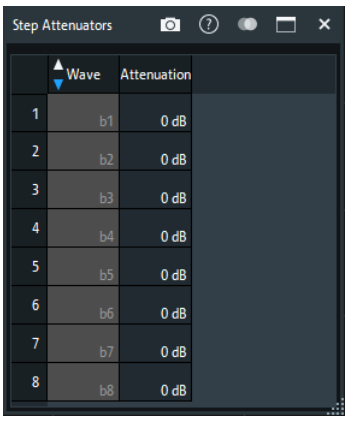


Step Attenuators
Defines an attenuation factor for the received waves at each analyzer port. At least one receiver step attenuator must be installed to use this feature; see Section 4.7.24, "Receiver step attenuators", on page 264.

Receiver step attenuators are used to adapt the received signal level to the input level range of the analyzer. They protect the instrument from being overloaded or damaged, e.g. if the DUT is a power amplifier.

Attenuation factors are port- and channel-specific. Possible values are 0 dB to 30 dB in steps of 10 dB. The analyzer rounds any entered value below the maximum attenuation to the closest step.

If the number of test ports (on the VNA and connected switch matrices) is higher than 4, the "Step Attenuators" input fields are hidden. They are replaced by a "Step Attenuators..." button that opens the following dialog:



Remote command:
`[SENSe<Ch>:]POWer:ATTenuation`

5.9.3 Bandwidth tab

Sets the measurement bandwidth and the shape of the digital IF filter for the active channel.

A system error correction (calibration) remains valid when the filter settings are changed.

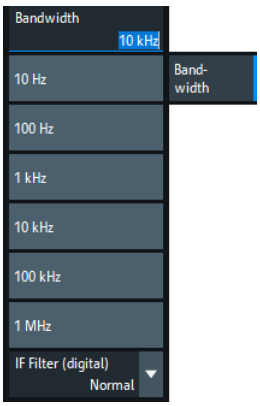
Optimizing the filter settings

A small bandwidth and a high selectivity of the digital IF filter suppress the noise level around the measurement frequency, and thus increase the dynamic range. However, the measurement time increases with smaller filter bandwidths and high selectivity. For small bandwidths, the filter settling time, which is inversely proportional to the bandwidth, is responsible for the predominant part of the measurement time.

The characteristics of the high selectivity filter make it particularly suitable for isolating unexpected spurious responses or known mixer products.

Segmented sweeps

In segmented frequency sweeps, the filter settings can be selected independently for each segment. See Section 5.10.2.2, "Define Segments dialog", on page 545.



Bandwidth
"Bandwidth" the measurement bandwidth of the IF filter. Within the value range, the entered value is rounded up to $1 \cdot 10^n$ Hz, $1.5 \cdot 10^n$ Hz, $2 \cdot 10^n$ Hz, $3 \cdot 10^n$ Hz, $5 \cdot 10^n$ Hz, $7 \cdot 10^n$ Hz ($n \geq 0$). Values exceeding the maximum bandwidth are rounded down.

The regular bandwidth range is 1 Hz to 1 MHz. Option R&S ZNB-K17 / R&S ZNBT-K17 enables bandwidths up to 10 MHz (see Section 4.7.4, "Receiver bandwidth 10 MHz", on page 243).

Remote command:
`[SENSe<Ch>:]BANDwidth[:RESolution]`
`[SENSe<Ch>:]BWIDth[:RESolution]`

IF Filter (digital)
Selects the shape of the IF filter:
- Filters with "Normal" selectivity provide the shortest settling time (recommended for fast measurements).
- "Medium" selectivity filters have steeper edges but require longer settling times.
- "High" selectivity filters have the steepest edges and the longest settling times. This filter type is suitable for isolating adjacent signals which are separated by a small frequency spacing.

Remote command:
`[SENSe<Ch>:]BANDwidth[:RESolution]:SELect`
`[SENSe<Ch>:]BWIDth[:RESolution]:SELect`

5.9.4 Average tab

Defines the number of consecutive sweeps to be averaged and enables/disables the sweep average.

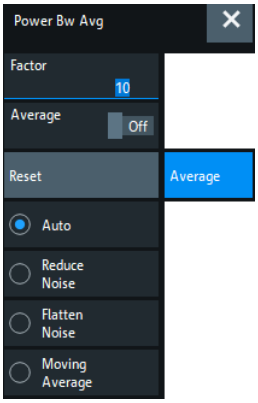
Effects of sweep averaging, alternative settings

An average over several sweeps reduces the influence of random effects in the measurement and therefore minimizes the noise level. The effect increases with the average factor, however, obtaining an averaged result requires several sweeps and therefore increases the measurement time.

Smoothing is an alternative method of compensating for random effects on the trace by averaging adjacent measurement points. Compared to the sweep average, smoothing does not significantly increase the measurement time but can eliminate narrow peaks and thus produce misleading results.

The sweep average is not frequency selective. To eliminate a spurious signal near the measurement frequency, alternative techniques (e.g. a smaller filter bandwidth) must be used.

The average factor is also valid for calibration sweeps. The calculation of system correction data is based on the averaged trace.



Factor / On / Reset

"Factor" defines the number of averaged traces, "On" enables or disables the sweep average, "Reset" starts a new average cycle. The average cycle is also restarted when the averaging mode is changed.

Remote command:

```
[SENSe<Ch>:]AVERage:COUNt
[SENSe<Ch>:]AVERage[:STATe]
[SENSe<Ch>:]AVERage:CLEar
```

Mode

Selects one of the following averaging algorithms:

- **"Auto":** Automatic selection between **"Reduce Noise"** and **"Flatten Noise"** mode, depending on the trace type.
- **"Reduce Noise":** Cumulative moving averages of the real and imaginary parts of each measurement result, provides the most effective noise suppression for the "Real" and "Imag" formats and for polar diagrams.

- **"Flatten Noise":** Cumulative moving averages of the (linear) magnitude and phase values, provides the most effective noise suppression for the "dB Mag", "Phase", "Unwr. Phase", and "Lin Mag" formats.
- **"Moving Average":** Simple moving averages of the real and imaginary parts of each measurement result; similar to "Reduce Noise", but with finite history.

Changing the mode resets the average cycle.

Note: For frequency conversion measurements, always "Flatten Noise" is used.

Remote command:
`[SENSe<Ch>:]AVERage:MODE`

5.10 Sweep Softtool

The "Sweep" softtool allows you to define the scope of the measurement in the active channel.

The available settings comprise the sweep type (with related parameters), the trigger conditions, and the periodicity of the measurement.

Access: Channel – [Sweep] hardkey

Background information

Refer to the following sections:

- Section 4.1.4, "Sweep control", on page 82
- Section 4.1.4.4, "Stimulus and sweep types", on page 87

5.10.1 Sweep Params tab

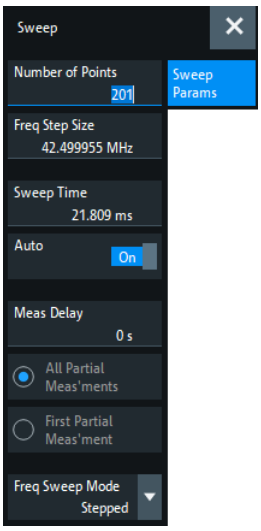
Allows you to define the scope and timing of the measurement in the active channel.

Segmented sweeps

In "Segmented" linear frequency sweeps, the sweep parameters can be set independently for each segment. See Section 5.10.2.2, "Define Segments dialog", on page 545.

System error correction

In general, the system error correction is no longer valid after a change of the sweep parameters. The status of the calibration is shown in the trace list. If the number of points is changed, the analyzer interpolates the correction data. The calibration label "Cal Int" is displayed. See also Section 4.5.4, "Calibration state labels", on page 175.



Number of Points
Sets the total number of measurement points per sweep. The minimum number of points is 1 (measurement at a single frequency/power/time value), the maximum number for frequency sweeps is 100001.

Together with the sweep range defined in the Stimulus tab of the "Stimulus" softtool, this parameter defines the grid of sweep points. The step size between two consecutive sweep points is constant on a linear scale (sweep types "Lin Freq", "Time" and "CW Mode") or on a logarithmic scale (sweep types "Log Freq" and "Power").

The sweep points for linear frequency sweeps can also be defined using the Freq Step Size.

Tip: Measurement time and screen resolution

Increasing the number of sweep points improves the resolution of the trace but increases the measurement time. The overall measurement time is composed of the hardware settling time at the beginning of the sweep plus the sum of the measurement times at each individual sweep point. Hence the measurement time increases roughly linearly with the number of points.

See also Section 4.1.4, "Sweep control", on page 82.

Remote command:
`[SENSe<Ch>:]SWEep:POINts`

Freq Step Size
The distance between two consecutive sweep points of a linear frequency sweep.

When you set "Freq Step Size", then first the "Number of Points" is adjusted to

max { p | "Start Frequency" + (p-1) · "Freq Step Size" ≤ "Stop Frequency"}

and then the "Stop Frequency" is adjusted to

"Start Frequency" + ("Number of Points"-1) · "Freq Step Size".

"Center Frequency" and "Span Frequency" are adjusted accordingly.

Note:

- This setting is valid for linear frequency sweeps only. It does not apply to logarithmic and segmented sweeps, power, time or CW mode sweeps.
- Decreasing the "Freq Step Size" generally increases the measurement time.

Remote command:
`[SENSe<Ch>:]SWEep:STEP`

Sweep Time / Auto
Varies the measurement time for a sweep or delays the start of each sweep.

- "Sweep Time" is the total measurement time for the sweep. The minimum possible sweep time is equal to the estimated value in "Auto" mode. Setting a larger sweep time is equivalent to defining a Meas Delay before each partial measurement.
- "Auto" minimizes the sweep time. The Meas Delay is set to 0 s. "Sweep Time" indicates the estimated sweep time, depending on the current measurement settings. The "Sweep Time" and "Meas Delay" values are maintained until changed explicitly if "Auto" is switched off.

If a time sweep is active, "Sweep Time" cannot be modified. The analyzer uses the previously defined sweep time settings.

In "Swept" frequency sweep mode, the "Sweep Time" is undefined. The "Sweep Time" field is cleared and disabled.

Note: For DUTs with large group delay, and with minimized sweep time, the data acquisition window of a transmission measurement can partly overlap with the signal from the previous sweep point. Do not use "Auto", but specify a positive "Meas Delay" in this case. To optimize for speed, start with a "Meas Delay" of 0, and increase it until the transmission traces become stable.

Remote command:
`[SENSe<Ch>:]SWEep:TIME`
`[SENSe<Ch>:]SWEep:TIME:AUTO`

Meas Delay
Adds a delay time before the start of every partial measurement. See Section 4.1.4.1, "Partial measurements and driving mode", on page 83.

"Meas Delay" is not available while a "Time" or "CW Mode" sweep is active.

As an alternative to increasing the delay (and thus the total sweep time), it is possible to select "Alternated" driving mode; see "Driving Mode" on page 699.

Remote command:
`[SENSe<Ch>:]SWEep:DWELl`

All Partial Meas'ments / First Partial Meas'ment
If Meas Delay is set to a value > 0, this setting allows you to define how the measurement delay is applied:

- If "All Partial Meas'ments" is selected, the delay time is added before each partial measurement. For a complete 2-port S-parameter measurement, the delay must be added twice per sweep point.
- If "First Partial Meas'ment" is selected, the delay time is added once per sweep point only, irrespective of the measured quantities and the number of partial measurements. The sweep time increases by the measurement delay times the number of sweep points.

Tip: A delay time before the start of each partial measurement increases the accuracy, in particular for measurements on DUTs with long settling times (e.g. quartz oscillators, SAW filters). Select "First Partial Meas'ment" if the DUT does not require an additional settling time due to the interchange of source and receive ports.

Remote command:
`[SENSe<Ch>:]SWEep:DWELl:IPOint`

Freq Sweep Mode
For frequency sweeps, you can select "Stepped" or "Swept" mode (see Section 4.1.4.5, "Stepped vs. swept mode", on page 88).

For a "Segmented" sweep, this setting can be overridden by segment-specific sweep modes (see Section 5.10.2.2, "Define Segments dialog", on page 545).

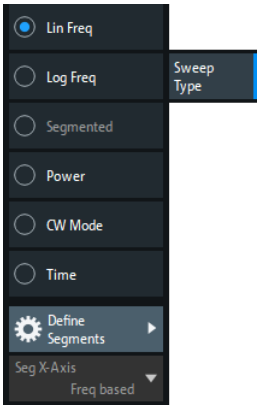
Note:
- "Swept" mode is **not** supported on the following instruments:
  - R&S ZNB43
  - Instruments equipped with synth boards SYNTH13DDS_DS and SYNTH13DDS_L.
    These synth boards are shipped with new R&S ZNBT8 (serial number ≥ 200000).
    See the hardware info to learn about the HW configuration of your instrument.
- Swept mode is only supported for linear frequency sweeps.
- While stepped mode is always possible, even for linear frequency sweeps there are channel setups that are not compatible with swept mode: an information popup displays the corresponding error code and description (see `[SENSe<Ch>:]SWEep:GENeration:ANALog:CONDition?`)
- It is essential to perform the calibration with the same "Freq Sweep Mode" settings as for the actual measurement; otherwise the calibration is deactivated ("Cal Off"). The Calibration Manager dialog provides concise information on possible mode mismatches.
- A scalar power calibration cannot be created or used in swept mode.

Remote command:
`[SENSe<Ch>:]SWEep:GENeration`
`[SENSe<Ch>:]SWEep:GENeration:ANALog:CONDition?`

5.10.2 Sweep Type tab

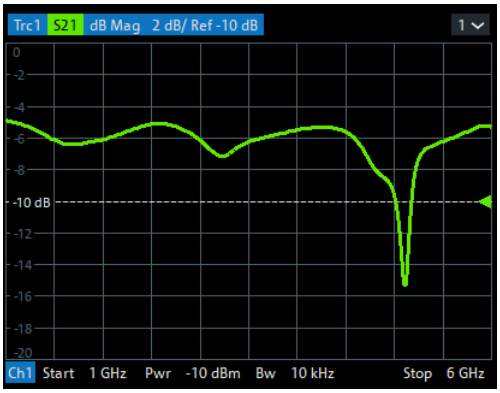
Defines the sweep variable (frequency/power/time) and the position of the sweep points across the sweep range.

5.10.2.1 Controls on the Sweep Type tab



Lin Freq
In a linear frequency sweep, the stimulus frequency is swept in equidistant steps over the continuous frequency range. The frequency range (sweep range) and the internal generator power can be specified in the "Stimulus" settings (see Section 5.8.1, "Stimulus tab", on page 526). The step width between two consecutive sweep points is constant and given by <Span>/(n - 1) where n is the specified "Number of Points" (n > 1).

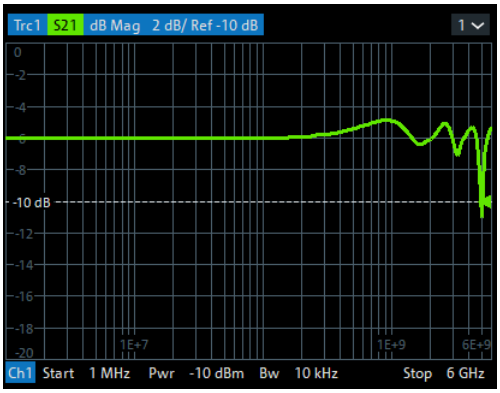
"Lin Freq" is the default sweep type. In a Cartesian diagram, the measurement result is displayed as a trace over a linear frequency scale (as known, e.g., from spectrum analyzers). The following example shows a "Lin Freq" sweep with the forward transmission parameter S_{21} as measured quantity, and a "dB Mag" scaled y-axis.



Remote command:
`[SENSe<Ch>:]SWEep:TYPE LINear`

Log Freq
In a "Log Freq" sweep, the stimulus frequency is swept on a logarithmic scale over the continuous frequency range. The frequency range (sweep range) and the internal generator power can be specified in the "Stimulus" settings (see Section 5.8.1, "Stimulus tab", on page 526). The sweep points are calculated from the "Span" and the specified "Number of Points" (n > 1) with the condition that the step width is constant on the logarithmic scale.

"Log Freq" sweeps are suitable for the analysis of a DUT over a large frequency range, e.g. over several octaves. In a Cartesian diagram, the measurement result is displayed as a trace over a logarithmic frequency scale. The following example shows a "Log Freq" sweep with the forward transmission parameter S_{21} as measured quantity, and a "dB Mag" scaled y-axis.



Tip: In "Log Freq" representation, limit lines and ripple limit lines appear as exponential curves; see Section 4.4.1.1, "Rules for limit line definition", on page 136.

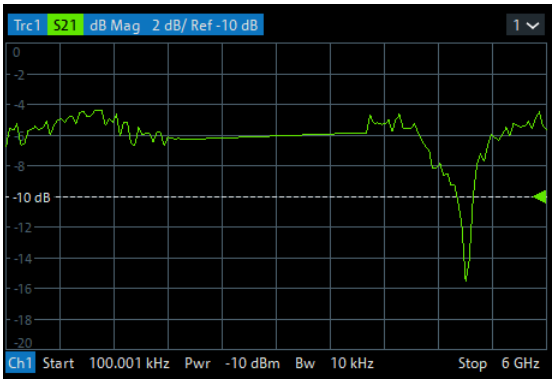
Remote command:
`[SENSe<Ch>:]SWEep:TYPE LOGarithmic`

Segmented
In a "Segmented" (linear) frequency sweep, the sweep range can be composed of several continuous frequency subranges or single frequency points. The subranges are termed sweep segments and are defined in the Define Segments dialog.

Sweep segments can overlap. The segment list must contain at least 2 distinct frequency points before a segmented frequency sweep can be started.

Instrument settings such as the internal generator power, the measurement (IF) bandwidth, and the frequency band of the local oscillator can be set independently for the individual segments.

Due to this flexibility, segmented frequency sweeps are suitable for any detailed analysis of a DUT at specified frequencies. In a Cartesian diagram, the measurement result is displayed as a trace over a linear frequency scale ranging from the lowest to the highest frequency point of all segments. The following example shows a segmented frequency sweep with 2 segments. The forward transmission parameter S_{21} is measured, and a "dB Mag" scaled y-axis is used. In the frequency range between the sweep segments, the trace is displayed as a straight line.



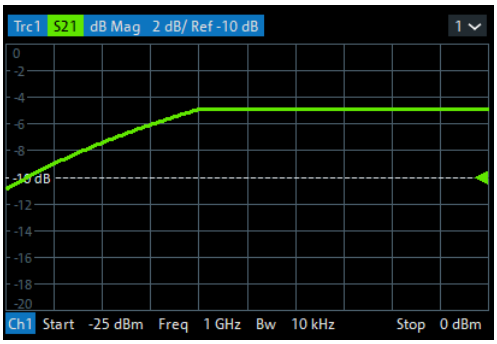
Tip: You can change to point based x-axis to improve the display of a segmented frequency sweep (see "Seg X-Axis" on page 544).

Remote command:
`[SENSe<Ch>:]SWEep:TYPE SEGMent`

Power
In a "Power" sweep, the internal generator power is swept in dB-linear, equidistant steps over a continuous power range. The generator power range (sweep range) and the fixed frequency can be specified in the "Stimulus" settings (see Section 5.8.1, "Stimulus tab", on page 526).

"Power" sweeps are particularly suitable for the analysis of nonlinear effects (saturation, compression) on active and passive DUTs (e.g. power amplifiers, mixers).

In a Cartesian diagram, the measurement result is displayed as a trace over a dB-linear power scale. The following example shows a "Power" sweep in the source power range between –25 dBm and 0 dBm, performed at a CW frequency of 1 GHz.



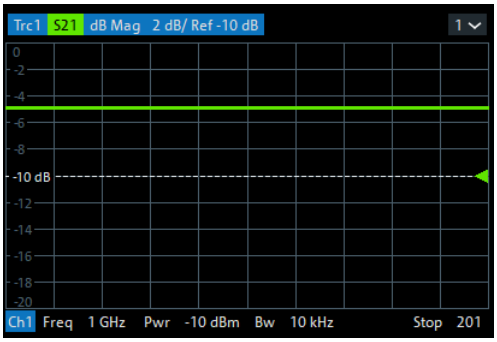
Any generator power calibration is included.

Remote command:
`[SENSe<Ch>:]SWEep:TYPE POWer`

CW Mode
"CW Mode" sweeps, like Time sweeps, are performed at constant frequency and stimulus power, which can be specified in the "Stimulus" settings (see Section 5.8.1, "Stimulus tab", on page 526).

The measurement is triggered according to the current trigger settings (see Section 5.10.3, "Trigger In tab", on page 552). Each trigger event triggers the first partial measurement of a measurement point. The time interval between two consecutive measurements depends on the trigger settings and the sweep parameters (especially the number of points). Any trigger mode is allowed.

A "CW Mode" sweep corresponds to the analysis of a signal over the time with a time scale and resolution that is determined by the trigger events. In a Cartesian diagram, the measurement result is displayed as a trace over a linear time scale (like, e.g., in an oscilloscope). The diagram is similar to the "Time" diagram. The following example shows a "CW Mode" sweep with a DUT that does not markedly change its transmission characteristics over the time.



Tip: Sweep time

The time interval between two consecutive trigger pulses must not be smaller than the minimum measurement time per measurement point. See "Sweep Time / Auto" on page 538.

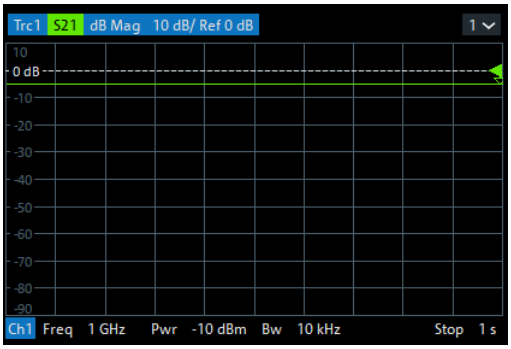
Remote command:

```
[SENSe<Ch>:]SWEep:TYPE POINt
```

Time

"Time" sweeps, like CW Mode sweeps, are performed at constant frequency and stimulus power, which can be specified in the "Stimulus" settings (see Section 5.8.1, "Stimulus tab", on page 526). A single sweep extends over a specified period of time, defined via the "Stop Time" setting. The time intervals between two consecutive sweep points are calculated according to "Stop Time"/(n - 1) where n is the selected Number of Points.

A "Time" sweep corresponds to the analysis of a signal over the time; the function of the analyzer is analogous to an oscilloscope. In a Cartesian diagram, the measurement result is displayed as a trace over a linear time scale. The following example shows a "Time" sweep with a DUT that does not markedly change its transmission characteristics over the time.



Tip: Sweep time

The minimum sweep time depends on the number of measurement points, the measurement bandwidth, the delay time before each partial measurement and the number of partial measurements required for each measurement point. The analyzer estimates this time, based on the current measurement settings.

If the "Stop Time" is smaller than the estimated minimum sweep time, the entered value is increased automatically.

Equidistance of sweep points

The analyzer tries to keep the time intervals between any two consecutive time sweep points equal: The time sweep samples are equidistant. Equidistance also holds for sweeps that range over several channels.

Remote command:

```
[SENSe<Ch>:]SWEep:TYPE CW
```

Define Segments

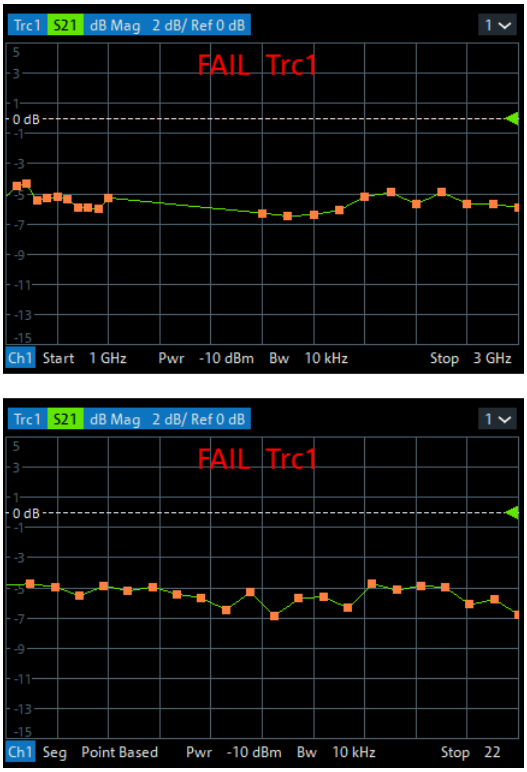
Opens the Define Segments dialog that allows to set up the channel for a Segmented frequency sweep.

Seg X-Axis

Scales the x-axis for a segmented frequency sweep:

- In "Freq based" mode, the x-axis covers the frequency ranges of all sweep segments, including possible gaps between the segments. Equal frequency spacings correspond to equal distances on the x-axis.
- In "Point based" mode, the x-axis shows all sweep points with equal spacings. Gaps between sweep segments are minimized; no diagram space is "wasted" on unused frequency ranges. Point based mode is indicated in the channel line.

The example below shows a segmented frequency sweep with two segments. The first segment ranges from 1 GHz to 1.4 GHz; the second segment from 2 GHz to 3 GHz. Both segments contain 11 sweep points. In point based mode (lower diagram), all sweep points are equidistant.



Tip: Overlapping limit line and ripple limit line segments are not displayed when a point based x-axis is active; see Section 4.4.1.1, "Rules for limit line definition", on page 136.

Remote command:
`[SENSe<Ch>:]FREQuency:SEGMent:AXIS`

5.10.2.2 Define Segments dialog

The "Define Segments" dialog defines all channel settings for a Segmented frequency sweep and imports or exports segmented sweep settings.

Access: Channel – [Sweep] > "Sweep Type" > "Define Segments"

The dialog contains a table to edit the individual segments of the sweep range. Use the Displayed Columns dialog to select the columns to be displayed and edited.

- Sweep segments are allowed to overlap.
- In contrast to standard frequency sweeps, the start and stop frequency in a sweep segment do not have to be different.
  So with a segmented sweep you can measure n_1 points at frequency f_1 (in segment 1), n_2 points at frequency f_2 (in segment 2) etc.

Controls in the Define Segments dialog

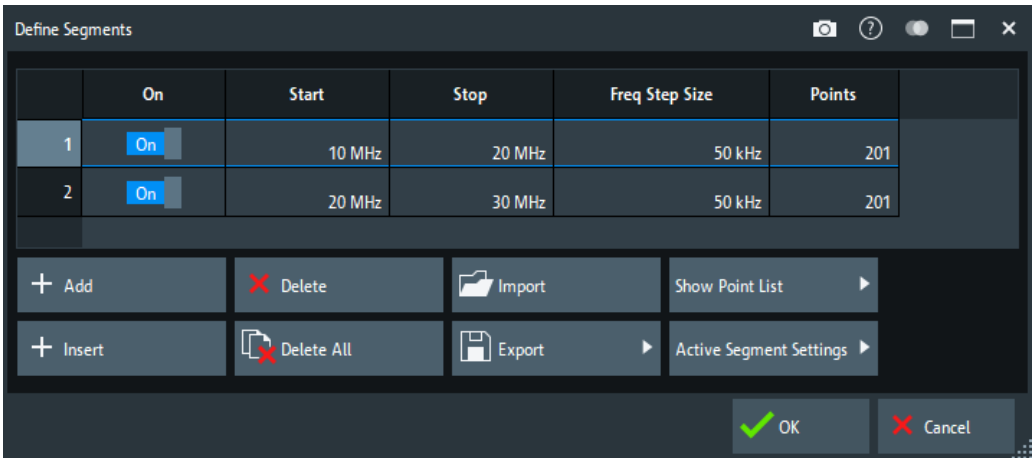


Table Columns

The table in the upper part of the "Define Segments" dialog contains an automatically assigned current number for each segment plus the following editable or non-editable columns:

- "On" provides controls to enable or disable each individual segment. Sweep points in inactive segments are not measured and not listed in the "Point List".
- "Start" is the stimulus (x-axis) value of the first point of the segment.
- "Stop" is the stimulus (x-axis) value of the last point of the segment.
- "Freq Step Size" is the distance between two consecutive sweep points of the segment.
  When you set this value, "Points" and "Stop" are adjusted accordingly (see "Freq Step Size" on page 537).
- "Points" is the number of sweep points in the segment. A single segment can consist of only one point. If "Points" is set to 1, then the "Stop" frequency is set equal to the "Start" frequency.

Note: Displayed Columns and Segment-specific Measurement Settings

The remaining columns allow you to replace channel-wide measurement settings by segment-specific ones. These columns are only displayed – and the corresponding segment-specific values are only applied – if they are selected in the Displayed Columns dialog.

Note: Limitations for overlapping segments

When overlapping sweep segments are created, the marker functions, trace evaluation functions, trace search functions and band filter functions are still available. It is possible, however, that these tools show an unexpected behavior when used in overlapping sweep segments. The reason is that the assignment of markers to traces in overlapping segments is ambiguous. If you want to analyze a particular segment using markers, turn "Off" all sweep segments that overlap with this segment.

Remote command:

```
[SENSe<Ch>:]SEGMent:COUNt?
[SENSe<Ch>:]SEGMent<Seg>[:STATe]
[SENSe<Ch>:]SEGMent<Seg>:FREQuency:STARt
[SENSe<Ch>:]SEGMent<Seg>:FREQuency:STOP
```

```
[SENSe<Ch>:]SEGMent<Seg>:FREQuency:STEP
[SENSe<Ch>:]SEGMent<Seg>:FREQuency:CENTer?
[SENSe<Ch>:]SEGMent<Seg>:FREQuency:SPAN?
[SENSe<Ch>:]SEGMent<Seg>:SWEep:POINts
```

Add / Insert / Delete / Delete All

The four buttons below the segment list extend or shorten the list.

- "Add" adds a new segment to the end of the list.
  The added segment covers a possible frequency gap between the preceding segment and the upper frequency limit of the analyzer. The "Start" frequency of the new segment is set equal to the "Stop" frequency of the preceding segment. The "Stop" frequency is equal to the upper frequency limit. The number of points is the same as the number of points of the preceding segment.
- "Insert" inserts a new segment before the active segment. The segment numbers of all segments after the new segment are incremented by one.
  The new segment covers a possible frequency gap between the two adjacent segments. A new segment which is inserted before segment no. 1 starts at the lower frequency limit of the analyzer. The number of points is the same as the number of points of the next segment.
- "Delete" removes the selected segment from the list.
- "Delete All" clears the entire segment list so it is possible to define or load a new segmented sweep range.

Remote command:

```
[SENSe<Ch>:]SEGMent<Seg>:ADD
[SENSe<Ch>:]SEGMent<Seg>:DEFine
[SENSe<Ch>:]SEGMent<Seg>:INSert
[SENSe<Ch>:]SEGMent<Seg>:DELete[:DUMMy]
[SENSe<Ch>:]SEGMent<Seg>:DELete:ALL
```

Show Point List

Opens a list of all active sweep points and their channel settings. All columns except "Point", "Segment" and "Frequency" are displayed only if they are explicitly selected; see "Displayed Columns dialog" on page 549.

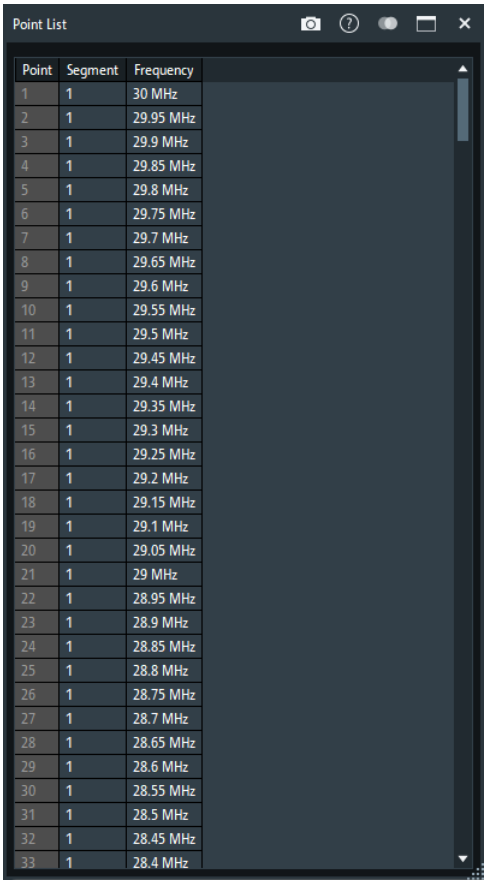


Figure 5-27: Point List

Import/ Export

The buttons open standard dialogs to import/export sweep segment settings to/from an ASCII file (`*.SegList`).

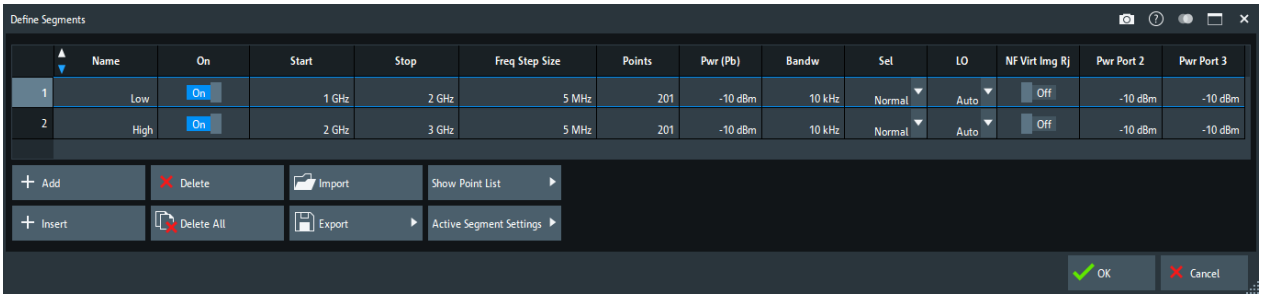
- "Import" replaces the current segment list by a sweep segment list loaded from a `*.SegList` file.
  Since FW V4.21 the R&S ZNB supports UTF8 files with and without BOM.
- "Export" stores the current sweep segments settings to a `*.SegList` file.

Sweep segment files

The analyzer uses a simple ASCII format to export sweep segment data. By default, the sweep segment file extension is `*.SegList`. The file starts with two comment lines containing the version and a third line reproducing the header of the segment list. The following lines contain the entries of all columns of the segment list, including the "Displayed Columns" that are configured in the "Define Segments" dialog.

Example:

The segmented sweep



is described by the following sweep segment file:

```
# Version 1.00
#
bo:State,str:Name,Start Frequency[MHz],Stop Frequency[MHz],int:No of Points,Source Power[dBm],IF Bandwidth[Hz]
true,Low,1.000000000000000E+003,2.000000000000000E+003,11,-1.000000000000000E+001,1.000000000000000E+004,Norma
true,High,2.000000000000000E+003,3.000000000000000E+003,11,-1.000000000000000E+001,1.000000000000000E+004,Norm
```

Note: The `*.SegList` file actually contains more columns listing all channel settings of the individual sweep segments. The headings of the additional columns are *IF Bandwidth [Hz]*, *en:IF Selectivity*, *en:IF Sideband*, *Meas Delay [µs]*, *bo:Sweep Time Auto*,*en:Frq Sweep Mode*.

Remote command:

`MMEMory:LOAD:SEGMent`

`MMEMory:STORe:SEGMent`

Displayed Columns dialog

The "Displayed Columns" dialog allows you to select the channel settings that can be defined per sweep segment. These settings are displayed in the Define Segments dialog and in the "Point List" (opened via Show Point List).

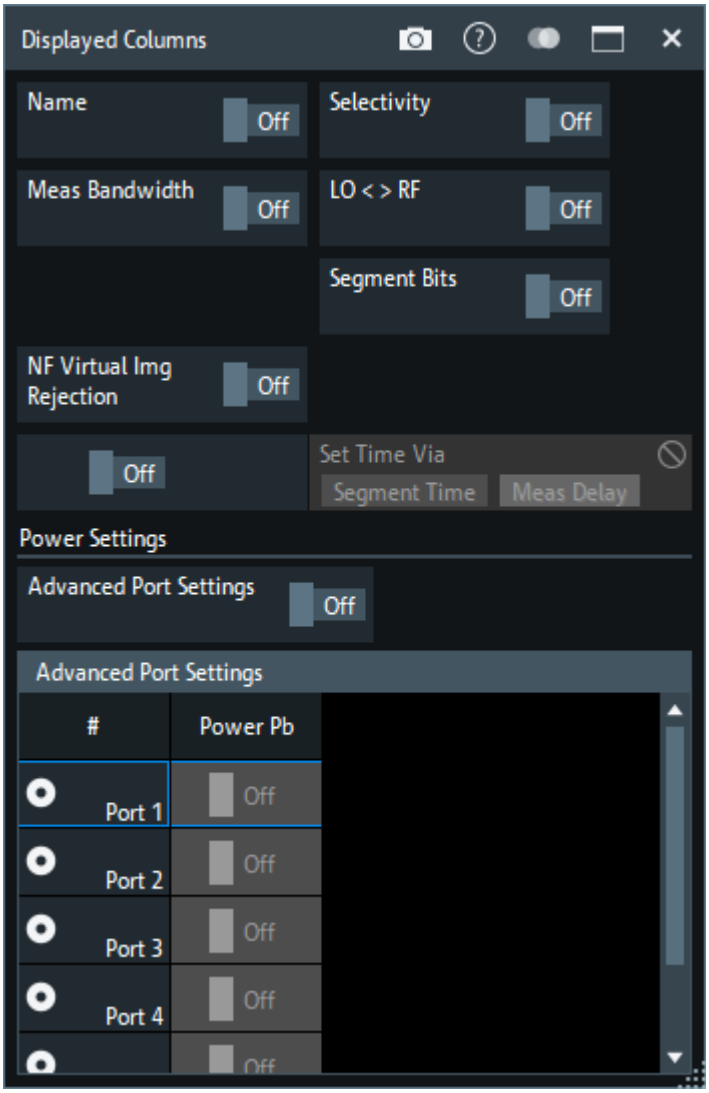
Access: Define Segments dialog > "Active Segment Settings"

All segment-specific settings can be modified in the "Define Segments" dialog. By default, the first sweep segment is created with the channel settings defined for unsegmented sweep types. When any further sweep segment created, it uses the channel settings of the previously active segment.

Related information

Refer to the following sections:

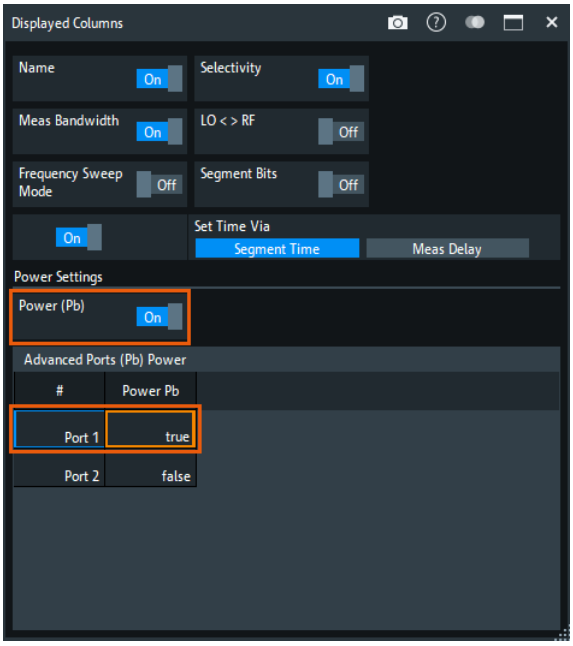
- Section 5.10.2.2, "Define Segments dialog", on page 545
- "Show Point List" on page 547



Optional Columns

Each selected (enabled) option adds a column to the segment list and the point list.

- "Name" allows you to assign a name to each segment. A segment name is a string that is allowed to contain letters, numbers and special characters.
- "Power (Pb)" allows you to define the internal source power (channel base power) for each individual sweep segment. See "Power" on page 531.
  If enabled, you have the possibility to define port-specific power levels by selecting the related ports in the "Advanced Ports (Pb) Power" table. Unselected ports are driven with the segment-specific power.



- "Meas Bandwidth" defines the IF filter bandwidth for each individual sweep segment. See "Bandwidth" on page 534.
- "NF Virtual Img Rejection" allows you to activate or deactivate virtual image rejection for individual sweep segments in noise figure channels. If the column is hidden, the channel-wide virtual image rejection setting is used.
  Enabling "NF Virtual Img Rejection" only takes effect if the channel is configured for Noise figure measurement, and if either "Noise Figure", "Gain" or "Noise Density" are measured.
- "Frequency Sweep Mode" defines whether the "Stepped" or "Swept" mode is used for the sweep segment (see Section 4.1.4.5, "Stepped vs. swept mode", on page 88).
  Swept mode is only supported for linear frequency sweeps. And even for linear frequency sweeps there are measurement setups that are incompatible with swept mode: an information popup displays the corresponding error code and description (see `[SENSe<Ch>:]SWEep:GENeration:ANALog:CONDition?`).
- "Selectivity" defines the selectivity of the IF filter used for each sweep segment. See "IF Filter (digital)" on page 534.
- "LO < > RF" allows you to define segment-specific "Image Suppr." settings; see "Image Suppr." on page 699.
- "Segment Bits" enables the definition of a segment-dependent 4-bit binary value to control four independent output signals at the USER PORT connector (lines 16, 17, 18, 19; see Section 13.3.1.1, "USER PORT", on page 1702). The output signals are 3.3 V TTL signals which can be used to differentiate between up to 16 independent analyzer states.
  For an application example, refer to the detailed remote control description (`OUTPut<Ch>:UPORt:SEGMent<Seg>[:VALue]`).
  Setting the segment bits does not change the analyzer state.
- "Time" defines the sweep time for each segment. The default configuration for a new segment is equal to the sweep time setting for unsegmented sweeps; see "Sweep Time / Auto" on page 538.

When "Time" is enabled, two new columns appear in the table. The first column is titled "Segment Time" or "Meas Delay", depending on the selected radio button below the "Time" control. The second column is titled "Auto" and is used to activate automatic sweep time setting.
"Segment Time" is the total measurement time for the sweep segment. The minimum segment sweep time to be set is equal to the estimated value in "Auto" mode.
"Meas Delay" sets a delay time allowing the DUT to settle before the hardware settings of the analyzer are changed and a new partial measurement is started.
Changing the "Meas Delay" modifies the "Segment Time" and vice versa.
"Auto" minimizes the sweep time. If "Auto" is selected for a segment, the columns "Segment Time" or "Meas Delay" (in the "Define Segments" dialog cannot be edited). "Segment Time" indicates the estimated sweep time, depending on the current measurement settings, the "Meas Delay" is 0 s. The segment sweep time and point delay values are maintained until changed explicitly if "Auto" is switched off.

Remote command:

```
[SENSe<Ch>:]SEGMent<Seg>:NAME
[SENSe<Ch>:]SEGMent<Seg>:BWIDth[:RESolution]:SELect:CONTrol
[SENSe<Ch>:]SEGMent<Seg>:BWIDth[:RESolution]:SELect
[SENSe<Ch>:]SEGMent<Seg>:BWIDth[:RESolution]:CONTrol
[SENSe<Ch>:]SEGMent<Seg>:BWIDth[:RESolution]
[SENSe<Ch>:]SEGMent<Seg>:LO
[SENSe<Ch>:]SEGMent:POWer[:LEVel]:CONTrol
[SENSe<Ch>:]SEGMent<Seg>:POWer[:LEVel]
[SENSe<Ch>:]SEGMent<Seg>:SWEep:GENeration
[SENSe<Ch>:]SWEep:GENeration:ANALog:CONDition?
[SENSe<Ch>:]SEGMent<Seg>:DEFine
[SENSe<Ch>:]SEGMent<Seg>:DEFine:SELect
[SENSe<Ch>:]SEGMent<Seg>:INSert
[SENSe<Ch>:]SEGMent<Seg>:INSert:SELect
[SENSe<Ch>:]SEGMent<Seg>:PORT<PortId>[:STATe]
[SENSe<Ch>:]SEGMent<Seg>:PORT<PortId>:POWer[:LEVel]
[SENSe<Ch>:]SEGMent<Seg>:SWEep:DWELl
[SENSe<Ch>:]SEGMent<Seg>:SWEep:DWELl:CONTrol
[SENSe<Ch>:]SEGMent<Seg>:SWEep:TIME
[SENSe<Ch>:]SEGMent<Seg>:SWEep:TIME:CONTrol
[SENSe<Ch>:]SEGMent<Seg>:SWEep:TIME:SUM?
OUTPut<Ch>:UPORt:SEGMent<Seg>:STATe
OUTPut<Ch>:UPORt:SEGMent<Seg>[:VALue]
CONTrol:AUXiliary:C[:DATA]
```

5.10.3 Trigger In tab

Selects the source of the trigger signal and provides additional trigger settings.

Trigger system of the analyzer

The trigger system is used to synchronize the analyzer's actions with events that can be provided by an internal or external signal or user-generated ("Manual"). Triggered

measurements are an alternative to the default mode ("None", "Continuous" sweep), where the measurement is continuously repeated without a fixed time reference.

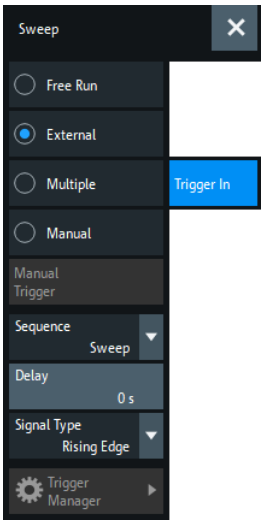
Any trigger event can start an entire sweep or a part of it. Moreover, it is possible to switch off the RF source between consecutive triggered measurement sequences, and to define a delay between trigger events and the measurement sequences.

Background information

Refer to Section 4.1.4.1, "Partial measurements and driving mode", on page 83.

5.10.3.1 Controls on the Trigger In tab

The trigger settings are also valid for calibration sweeps. Hence, in external trigger mode, the external trigger signal must be available during the system error correction, too. To start the calibration sweeps without delay, use the "FreeRun" trigger type.



The following buttons in the "Trigger In" tab open related dialogs:

- **"Trigger Manager":** Trigger Manager dialog

Free Run (None) / External / Manual / Multiple

These four buttons select the trigger source:

- In free run mode, a new measurement is started immediately without waiting for a trigger event and without a fixed time reference. The remaining trigger settings are not valid. In "Continuous" sweep mode, the measurement is repeated as fast as possible.
- In "External" trigger mode, the measurement is triggered by an external TTL signal, applied to one of the following rear panel connectors:
  - BNC connector Trigger In
  - Pin 2 of the USER PORT connector
  - Pin 18 of the Handler I/O (universal interface)

The trigger inputs are equivalent; no additional setting for signal routing is required. For detailed specifications of the trigger signals, refer to Section 13.3.1.1, "USER PORT", on page 1702.
The "External" trigger mode can be configured using the Sequence, Delay and Signal Type settings.

- In "Manual" trigger mode, the trigger signal is generated by the "Manual" button.
- If "Multiple" is active, the trigger sources for different triggered measurement sequences, the trigger slope, and the trigger delay can be selected individually using the Trigger Manager dialog.
  In particular, it is possible to use different external trigger sources.

Remote command:

```
TRIGger<Ch>[:SEQuence]:SOURce EXTernal
```

Manual

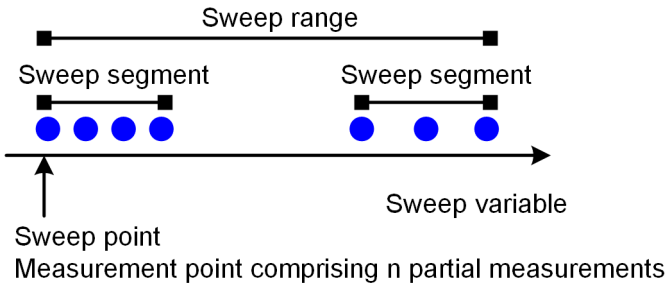
Generates the trigger event for "Manual" trigger mode and is disabled unless this mode is active.

Remote command:

```
TRIGger<Ch>[:SEQuence]:SOURce MANual
*TRG
```

Sequence

Selects the measurement cycle or sequence of actions to be triggered in "External" or "Manual" mode.



- "Sweep" means that each trigger event starts an entire sweep, according to the current sweep configuration.
- "Point" means that each trigger event starts the measurement at the next sweep point.
- "Partial Measurement" means that each trigger event starts the next partial measurement at the current or at the next sweep point. If every sweep point only requires a single partial measurement, this option is equivalent to "Point".
  See also Section 4.1.4.1, "Partial measurements and driving mode", on page 83.
- "Segment" means that each trigger event starts the next sweep segment within the current sweep. If a sweep type other than Segmented is active, this option is equivalent to "Sweep".

Trigger sequence and driving mode

Some trigger sequences are incompatible with the "Alternated" Driving Mode:

- For "Point" triggering, the VNA automatically switches to "Chopped" mode internally.

- For "Segment" triggering, the VNA switches to "Block Alternated" mode internally

Note: Adjust the trigger events to the triggered measurement sequence. If the analyzer receives a trigger event while the last sequence is still running, it skips the trigger event and generates a message.

Remote command:
`TRIGger<Ch>[:SEQuence]:LINK`

Delay
Specifies a delay time between the trigger event and the start of the next measurement sequence.

The specified "Delay" must be zero or positive, so that the trigger event precedes the start of the measurement (post-trigger).

If "Multiple" is active, the "Delay" can be selected individually using the Trigger Manager dialog.

Remote command:
`TRIGger<Ch>[:SEQuence]:HOLDoff`

Signal Type
Specifies the "External" trigger mode in detail.

- **"Rising Edge"/"Falling Edge"** means that the rising/falling slope of every external trigger pulse can trigger a single measurement sequence.
- **"Active High"/"Active Low"** means that the analyzer measures in freerun mode ("None") as long as the external trigger signal is high/low. The measurement is discontinued when the trigger signal changes to low/high.

Remote command:
`TRIGger<Ch>[:SEQuence]:SLOPe`

5.10.3.2 Trigger Manager dialog

The "Trigger Manager" dialog defines individual trigger sources and delays for the triggered measurement sequences. The dialog is available and its settings are valid if the analyzer is configured for "Multiple" (see "Free Run (None) / External / Manual / Multiple" on page 553).

Access: Channel – [Trigger] > "Trigger Manager"

Background information
Refer to Section 4.1.4.1, "Partial measurements and driving mode", on page 83.

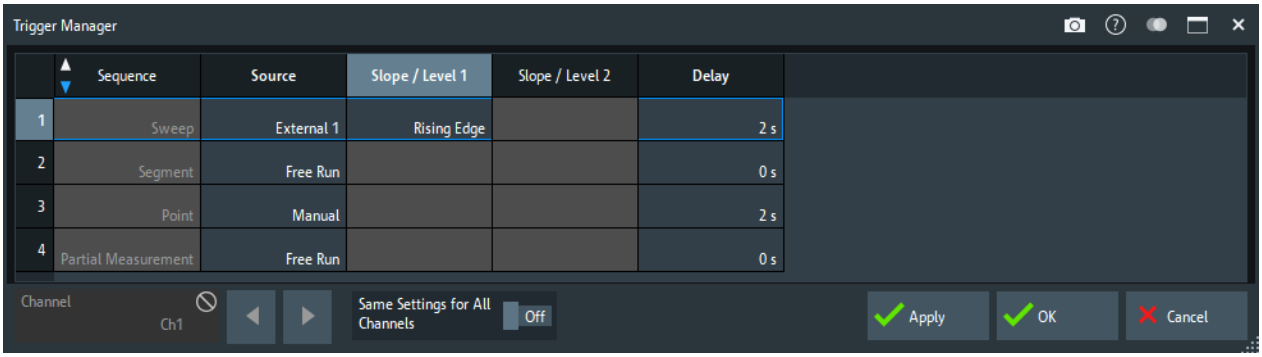


Figure 5-28: Example of a multiple trigger configuration

The table in the "Trigger Manager" dialog contains several editable (white) or non-editable (gray) columns. For a description, see the single trigger settings on the Trigger In tab:

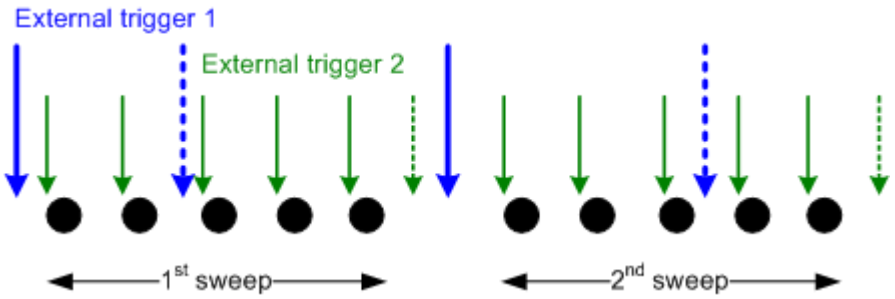
- "Sequence" on page 554
- "Source" → "Free Run (None) / External / Manual / Multiple" on page 553
  For the second trigger source and logical combinations of the two trigger sources, see "... |Source| ..." on page 556.
- "Slope / Level" → "Signal Type" on page 555
- "Delay" on page 555

... |Source| ...

The table defines all settings related to "Multiple" (see Section 5.10.3, "Trigger In tab", on page 552). For all measurement sequences the following trigger sources are available:

- "External 1" is the external trigger signal fed in at either the Trigger In connector on the rear panel or pin 2 of the USER PORT connector.
- "External 2" is the external trigger signal fed in at pin 25 of the USER PORT connector on the rear panel.
- "External 1 and 2"/"External 1 or 2" means that the measurement sequence is initiated after the analyzer has received an event from external trigger signal 1 and/or 2. Specify a Signal Type for both trigger signals.

For example, a new sweep can be triggered by an external trigger no. 1, while the individual sweep points are triggered by external trigger no. 2. External trigger 1 is ignored if a sweep is running, external trigger 2 is ignored if there is no running sweep. In the figure below a sweep comprises 5 measurement points and dotted arrows depict ignored trigger events.

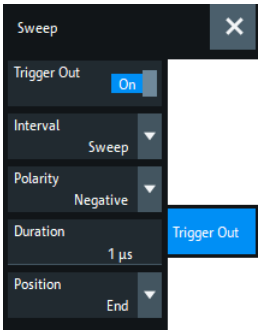


Remote command:
TRIGger<Ch>[:SEQuence]:MULTiple:SOURce
TRIGger<Ch>[:SEQuence]:MULTiple:SLOPe<Num>
TRIGger<Ch>[:SEQuence]:MULTiple:HOLDoff

5.10.4 Trigger Out tab

The R&S ZNB provides a configurable output trigger signal to synchronize external devices with the measurement. This signal is available at the rear panel connector Trigger Out.

The "Trigger Out" tab was introduced in firmware version 3.70. Before that, configuration of the output trigger signal was available via remote control only.



Trigger Out Active
Enables/disables the external trigger output EXT TRIG OUT for the active channel.

Remote command:
TRIGger:CHANnel<Ch>:AUXiliary[:ENABle]

Interval
Specifies whether the trigger output pulses are sent once every sweep, sweep segment, sweep point, or partial measurement.

Remote command:
TRIGger:CHANnel<Ch>:AUXiliary:INTerval

Polarity
Selects the polarity of the output trigger pulses.

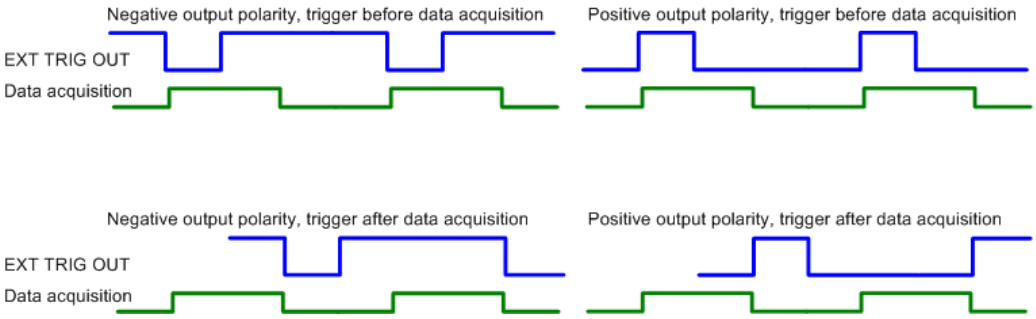


Figure 5-29: Output trigger signal at EXT TRIG OUT

Remote command:
`TRIGger:CHANnel<Ch>:AUXiliary:OPOLarity`

Duration
Specifies the width of the output trigger pulses at Trigger Out.

The trigger duration must be shorter than the selected sweep interval.

Remote command:
`TRIGger:CHANnel<Ch>:AUXiliary:DURation`

Position
Defines whether the trigger output pulse is generated at the start or at the end of the selected sweep interval. See Figure 5-29.

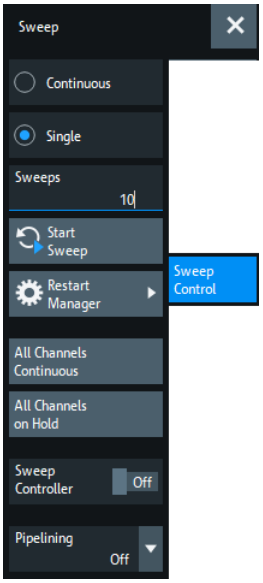
Remote command:
`TRIGger:CHANnel<Ch>:AUXiliary:POSition`

5.10.5 Sweep Control tab

Allows you to control the sweep execution.

The availability of certain settings depends on the selected Remote Language.

5.10.5.1 Controls on the Sweep Control tab



Continuous / Single
Activate either continuous (default) or single sweep mode.

- In "Continuous" sweep mode, the analyzer measures the related channel (or channels) continuously, repeating the current sweep over and over.
- In "Single" sweep mode, the measurement is stopped after the configured number of Sweeps.
  Restart Sweep initiates a new measurement cycle.

For the DEFAULT Remote Language, this setting applies to the active channel only. Use All Channels Continuous/All Channels on Hold to select a common sweep mode for all channels.

For remote languages other than DEFAULT, this setting applies to all channels.

Remote command:
`INITiate<Ch>:CONTinuous`
`[SENSe<Ch>:]SWEep:MODE`
See also:
`CONFigure:CHANnel<Ch>:MEASure[:STATe]`
`CONFigure:CHANnel:MEASure:ALL[:STATe]`

Sweeps
Defines the number of sweeps to be performed in "Single" sweep mode (see "Continuous / Single" on page 559).

This setting applies to the active channel. In "Continuous" sweep mode, it is not available (grayed out).

For remote languages other than DEFAULT, the "Sweeps" setting is disabled. You can define the number of sweeps in the Restart Manager dialog.

Remote command:
`[SENSe<Ch>:]SWEep:COUNt`
`[SENSe:]SWEep:COUNt:ALL`

Restart Sweep
Stops the current measurement cycle and starts a new one.

For the DEFAULT Remote Language, the effect of this command depends on the sweep modes of the active channel, and the restart behavior defined in the Restart Manager dialog:

	"Sweep Active Channel"	"Sweep All Channels"
Active channel in "Continuous" sweep mode	The "Continuous" sweep mode channels are swept cyclically, the "Single" sweep channels are on hold.	First the "Single" sweep mode channels are swept one after the other, each with its channel-specific number of "Sweeps". Then the "Continuous" sweep mode channels are sweep cyclically.
Active channel in "Single" sweep mode	First the active channel is swept with its channel-specific number of "Sweeps". Then the "Continuous" sweep mode channels are sweep cyclically.	

For remote languages other than DEFAULT, the effect of the "Restart Sweep" command is determined by the selected (common) sweep mode and restart behavior:

	"Sweep Active Channel"	"Sweep All Channels"
All channels in "Continuous" sweep mode	The channels are swept cyclically	
All channels in "Single" sweep mode	The active channel is swept "Sweeps" times. All other channels are on hold	The channels are swept one after the other, each of them "Sweeps" times.

Remote command:
`INITiate<Ch>[:IMMediate][:DUMMy]`
`INITiate[:IMMediate]:ALL`

Stop Sweep
Stops an ongoing sweep

Remote command:
`INITiate<Ch>:STOP`

Restart Manager
Opens the Restart Manager dialog.

All Channels Continuous/All Channels on Hold
Selects a common sweep mode for all channels of the active recall set.
- "All Channels Continuous": all channels are set to "Continuous" sweep mode
- "All Channels on Hold": all channels are set to "Single" sweep mode.
  Restart Sweep initiates a new measurement cycle.

These actions are only available for the DEFAULT Remote Language). For other remote languages, the Continuous / Single setting applies to all channels and the "All Channels Continuous"/"All Channels on Hold" buttons are grayed out;

Remote command:
`INITiate:CONTinuous:ALL`

Sweep Controller
Activates/deactivates the (resizable) "Sweep Info" dialog, which displays the current sweep stage. The "Sweep Info" dialog is particularly useful for long duration sweeps that are executed in single sweep mode: by observing the dialog, it is easy to realize when the sweep is done.

The possible sweep stages and how they are displayed partly depend on the selected sweep mode (see "Continuous / Single" on page 559):

Sweep stage	Sweep controller display	
	in "Continuous" sweep mode	in "Single" sweep mode
No ongoing sweep	*Idle*	*Idle*
Sweep is being prepared	*Preparing*	*Preparing*
Ongoing sweep [1]	*Continuous*	*Sweeping* if #Sweeps = 1
		Sweeping m/n if n = #Sweeps > 1
Measurement results are being calculated	*Continuous calculation*	*Calculation*
[1] The VNA is sweeping or waits for a trigger signal.		

Pipelining
Allows you to activate pipelining of measurement data.

This applies to all channels in the active setup.

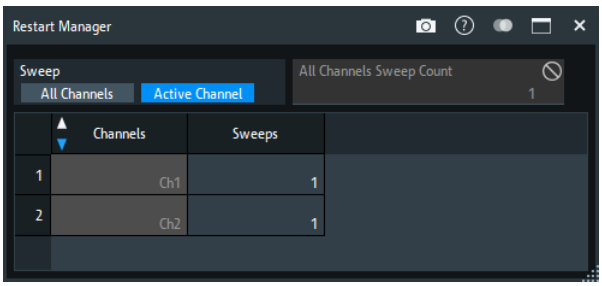
Remote command:
`INITiate:PIPelining:STATe`
(`CALCulate:PIPelining:DATA?`)

5.10.5.2 Restart Manager dialog

The "Restart Manager" dialog defines the Restart Sweep behavior in "Single" sweep mode.

Access: Channel – [Sweep] > "Sweep Control" > "Restart Manager"

Related information
Refer to Section 4.1.4, "Sweep control", on page 82.



Sweep All Channels/Sweeps
If you select "Sweep All Channels", Restart Sweep restarts the sweep in all channels of the active recall set.

For the DEFAULT Remote Language, the number of sweeps in single sweep mode is channel-specific and can be defined in the table below the separator line. The "Sweeps" column reflects the Sweeps settings of the individual channels. The "Sweeps" setting above the separator line is unused and disabled.

For remote languages other than DEFAULT, the "Sweeps" setting above the separator line defines the common number of sweeps that are executed on each channel in single sweep mode. The table below the separator line is unused and disabled.

Tip: In remote control, it is possible to retrieve the results acquired in any of the sweeps within a single sweep group.

Remote command:
```
INITiate[:IMMediate]:ALL
INITiate<Ch>[:IMMediate]:SCOPe ALL
[SENSe<Ch>:]SWEep:COUNt
```

Sweep Active Channel/Sweeps (table)
If you select "Sweep Active Channel", then in single sweep mode Restart Sweep only restarts the sweep in the active channel.

The "Sweeps" column reflects the Sweeps settings of the individual channels.

Remote command:
```
INITiate<Ch>[:IMMediate]:SCOPe SINGle
[SENSe<Ch>:]SWEep:COUNt
```

5.11 Cal softtool

The "Cal" softtool provides all functions related to system error calibration, scalar power calibration, and "SMARTerCal".

Access: Channel – [Cal]

5.11.1 Start Cal tab

The "Start Cal" tab provides access to all functions for automatic or manual calibration. Calibration of the R&S ZNB is a fully guided process.

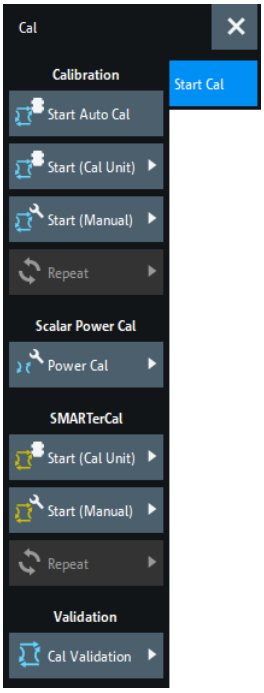
Background information

Refer to the following sections:

- Section 4.5, "Calibration", on page 154
- Section 4.5.5, "Automatic calibration", on page 175
- Section 4.5.6, "Scalar power calibration", on page 187
- Section 4.5.7, "SMARTerCal", on page 193
- Section 4.1.5, "Data flow", on page 89

It is possible to set up, perform and activate multiple system error corrections or SMARTerCals in a single calibration procedure. However, this Multiple Cal in Calibration Wizard feature is disabled by default and has to be activated in the "Calibration" tab of the "System Config" dialog.

5.11.1.1 Controls on the Start Cal tab



- The "Start Auto Cal" and "Start (Cal Unit)" buttons are only active if a calibration unit is connected to the analyzer.
- If there is more than one channel in the current recall set at the time you initiate a calibration, the Channel Selection for Calibration dialog is opened.

Start Auto Cal

Performs a fully automatic system error calibration for the active channel, using the active calibration unit with its factory calibration and auto-detection of ports.

If one port is active, an OSM calibration is performed. If two ports are active, an UOSM calibration is performed.

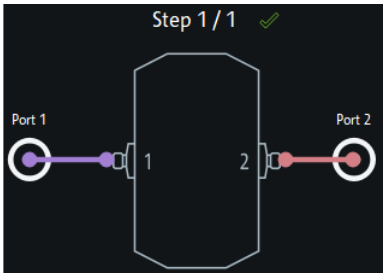
Calibration – Start (Cal Unit)
Depending on the Multiple Cal in Calibration Wizard setting, the "Start (Cal Unit)" button either opens the Calibration Unit wizard or the Calibration Unit wizard (MultiCal).

Calibration – Start (Manual)
Depending on the Multiple Cal in Calibration Wizard setting, the "Start (Manual)" button either opens the Calibration Setting wizard or the Calibration Setting wizard (MultiCal).

Calibration – Repeat
Re-enters the data acquisition step of the current calibration, restoring the related channel settings. This provides a convenient way to repeat or correct an existing calibration without necessarily repeating all measurements.

During recalibration, a hollow green checkmark indicates that the corresponding calibration sweep was not yet repeated:





Scalar Power Cal – Power Cal
Opens the Power Cal wizard.

If the active recall set contains several channels, an "Info" box lets you decide whether the R&S ZNB acquires calibration data for the active channel or for all channels.

SMARTerCal – Start (Cal Unit)
Depending on the Multiple Cal in Calibration Wizard setting, the "Start (Cal Unit)" button either opens the SMARTerCal wizard (Cal Unit) or the SMARTerCal wizard (Cal Unit, MultiCal).

SMARTerCal – Start (Manual)
Depending on the Multiple Cal in Calibration Wizard setting, the "Start (Cal Unit)" button either opens the SMARTerCal wizard (manual) or the SMARTerCal wizard (manual, MultiCal).

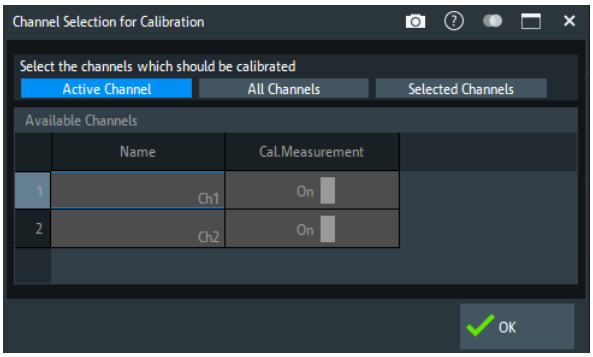
SMARTerCal – Repeat
Reenters the data acquisition step of the current SMARTerCal, restoring the related channel settings. This function provides a convenient way to repeat or correct an existing SMARTerCal without necessarily repeating all measurements.

Cal Validation ...
Opens the Cal Validation dock widget that allows you to set up and run a Calibration validation.

If more than one calibration unit is connected, an additional dialog allows you to select the appropriate calu, along with its characterization and ports to be used.

5.11.1.2 Channel Selection for Calibration dialog

If you initiate a calibration and there is more than one channel in the current recall set, you are prompted to select the channels to be calibrated:



Use

- "Active Channel" if you only want to calibrate the active channel
- "All Channels" if you want to calibrate all channels in the current recall set
- "Selected Channels" if you want to calibrate only certain channels

5.11.1.3 Calibration Unit wizard

The "Calibration Unit" wizard guides you through the setup and execution of an automatic calibration.

Access: Channel – [Cal] > "Start Cal" > Calibration – "Start (Cal Unit)"

The wizard proceeds in three steps.

1. Step 1: Ports allows you to configure the calibration and to select a calibration unit (along with its characterization).
2. Step 2: Connections allows you to define the port assignments between the R&S ZNB and the calibration unit.
3. Step 3: Cal Unit guides you through the required measurements on the standards provided by the calibration unit. Finally, you can either apply the resulting error terms to the related channels, or discard them.

Background information

Refer to Section 4.5.5, "Automatic calibration", on page 175.

- A successful calibration supersedes the previous calibration, discarding all previous system error correction data.
  To keep older correction data, you can transfer them to the calibration "Pool" using the Calibration Manager dialog.
- The system error correction data that are determined in a calibration procedure, are stored in the analyzer. It is possible to retrieve these data using the remote control command `[SENSe<Ch>:]CORRection:CDATa`.
  If external switch matrices are involved, use `[SENSe<Ch>:]CORRection:SMATrix:CDATa` instead.

Step 1: Ports

Allows you to define the calibration to be performed and to select the calibration unit to be used (along with its characterization).

Background and related information

Refer to the following sections:

- Section 4.5.1, "Calibration types", on page 155
- Section 4.5.5.3, "Characterization of calibration units", on page 179
- "Step 1: Ports" on page 602 (for SMARTerCal)

The layout of the main panel depends on the number of test ports available:

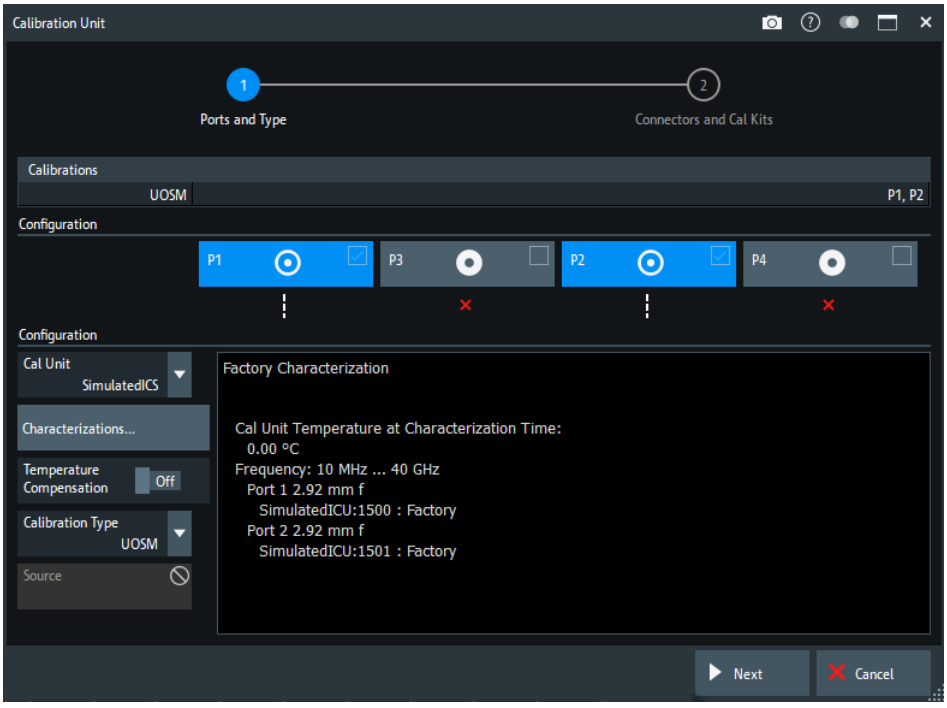


Figure 5-30: Calibration Unit wizard, step 1: Ports

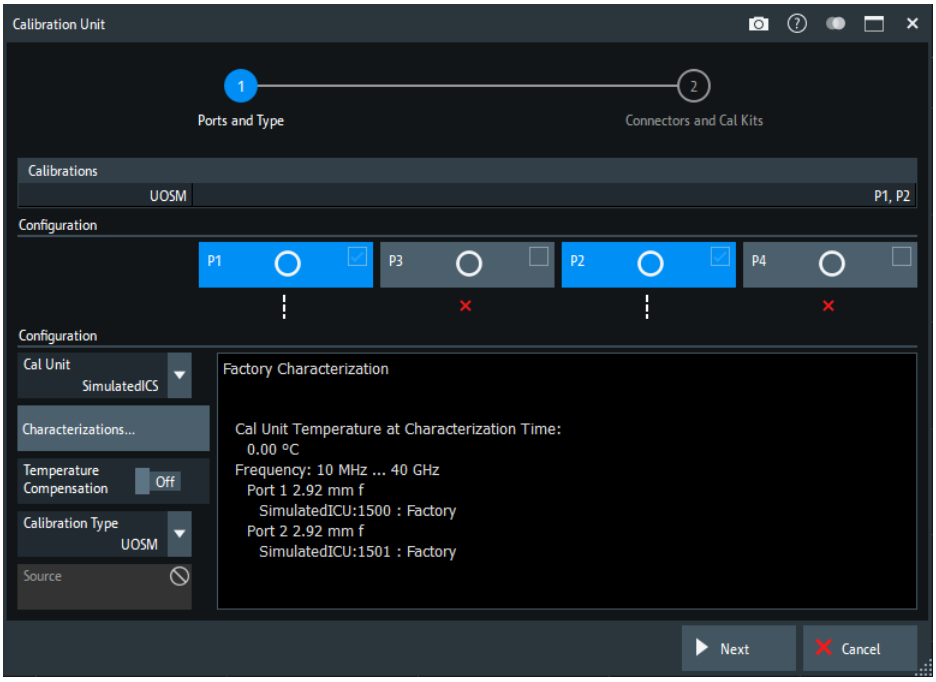


Figure 5-31: Calibration Unit wizard, step 1: Ports (inline calibration system)

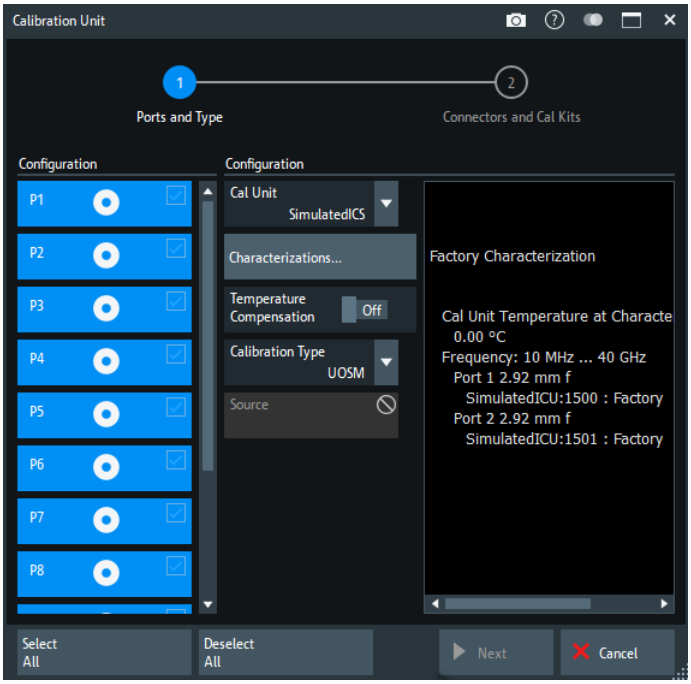


Figure 5-32: Calibration Unit wizard, step 1: Ports (> 4 test ports)

Ports
Selects the test ports to be calibrated.

Note: Calibration and port de-/activation.

The analyzer firmware automatically activates/deactivates ports during/after a (successful) calibration:

- Calibrated ports that were previously disabled, are automatically enabled as single-ended logical ports.
- An uncalibrated port that is not used by a measurement (i.e. the port is not required by any trace of the related channel) is disabled.

Remote command:
The port parameters in many calibration commands define the calibrated ports.

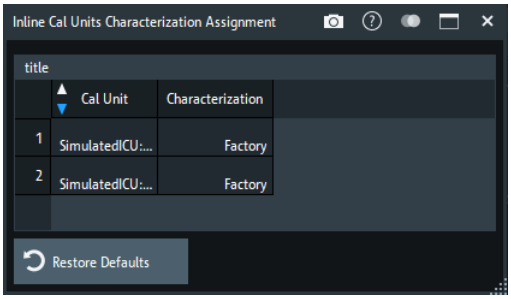
Cal Unit
Displays the connected calibration units. The R&S ZNB auto-detects all calibration units which are connected to one of its USB ports. If several cal units are connected, one of them must be selected for calibration (active cal unit). A warning is displayed if the current sweep range of the network analyzer exceeds the characterized frequency range of the calibration unit.

For background information, see Section 4.5.5, "Automatic calibration", on page 175.

Remote command:
```
SYSTem:COMMunicate:RDEVice:AKAL:ADDRess:ALL?
SYSTem:COMMunicate:RDEVice:AKAL:ADDRess
```

Characterization
Displays all characterizations that are stored in the active cal unit. The "Factory" characterization is available for all calibration units; it ensures an accurate calibration for all standard applications. To account for modifications of the cal unit such as the connection of additional adapters, you can generate modified sets of characterization data using the cal unit characterization wizard. See Section 5.11.2.3, "Characterize Cal Unit dialog", on page 643. By default, the R&S ZNB uses the last generated cal unit characterization.

For an inline calibration system, each connected inline calibration unit has its own set of characterizations:



Tip: If the characterization wizard is password-protected, the "Characterization" button is unavailable. Use this functionality to prevent inadvertent activation of inappropriate characterizations. See "Authentication" on page 644.

See also Section 4.5.5.3, "Characterization of calibration units", on page 179.

Remote command:
```
[SENSe<Ch>:]CORRection:COLLect:AUTO:CONFigure
SYSTem:COMMunicate:RDEVice:AKAL:CKIT
SYSTem:COMMunicate:RDEVice:AKAL:CKIT:CATalog?
```

SYSTem:COMMunicate:RDEVice:AKAL:CKIT:STANdard:CATalog?
SYSTem:COMMunicate:RDEVice:AKAL:SDATa?
Query further cal unit properties:
SYSTem:COMMunicate:RDEVice:AKAL:DATE?
SYSTem:COMMunicate:RDEVice:AKAL:FRANge?
SYSTem:COMMunicate:RDEVice:AKAL:PORTs?
SYSTem:COMMunicate:RDEVice:AKAL:WARMup[:STATe]?

Temperature Compensation
Available for inline calibration systems (ICS) only. Applies to all inline calibration units (ICUs) that are connected to the ICS and that are used with factory characterization data.

If enabled, the R&S ZNB firmware adjusts the factory characterization data according to the ICU temperatures.

Remote command:
[SENSe<Ch>:]CORRection:COLLect:AUTO:TEMPcomp

Calibration Type / Source
Selects the calibration type for the selected physical ports. For an overview, refer to Table 4-7.

The reflection calibration types can be used for any combination of physical ports: reflection calibrations are performed for each selected port.

A transmission calibration type requires at least two physical ports. For the **unidirectional** transmission calibration types ("Trans Norm", "One Path Two Ports"), the direction (**"Source"** port) must be specified in addition.

Transmission normalization and "One Path Two Ports" calibration types require two-port (Through) characterization data for the cal unit. For current calibration units, however, these two-port characterizations are typically not part of the factory characterization. If a tooltip indicates missing two-port characterization data, simply perform a new characterization of your cal unit. In the first dialog of the "Characterization" wizard, select "Take All OSM and Through" to make sure that the necessary two-port data is acquired. See also "Characterization wizard" on page 645.

Remote command:
[SENSe<Ch>:]CORRection:COLLect:AUTO:TYPE
[SENSe<Ch>:]CORRection:COLLect:AUTO:PORTs:TYPE
[SENSe<Ch>:]CORRection:COLLect:AUTO:CONFigure

Common Port
This combo-box is only available if Use Reduced Number of Throughs for Cal Unit is activated in the "Calibration" tab of the "System Config" dialog.

It defines the port that must appear in each port assignment of an automatic calibration, and that will be used as the "center" of all Through measurements.

Remote command:
[SENSe<Ch>:]CORRection:COLLect:AUTO:CPORt

Select All/Deselect All
Selects/deselects all ports.

These buttons are only shown if more than 4 ports are available.

Next
Proceeds to Step 2: Connections. "Next" is unavailable (and a warning is displayed) if the following happens:

- The selected characterization data do not cover all the ports to be calibrated.
- The selected characterization data do not contain all standards needed for the selected calibration type.

Step 2: Connections

Defines the port assignments between the R&S ZNB and the calibration unit.

Background information
Refer to:

- Section 4.5.5.6, "Multiple port assignments", on page 185 for details on multiple port assignments
- "Step 2: Connections" on page 606 for SMARTerCal.

Depending on the number of test ports, the user interface is slightly different:

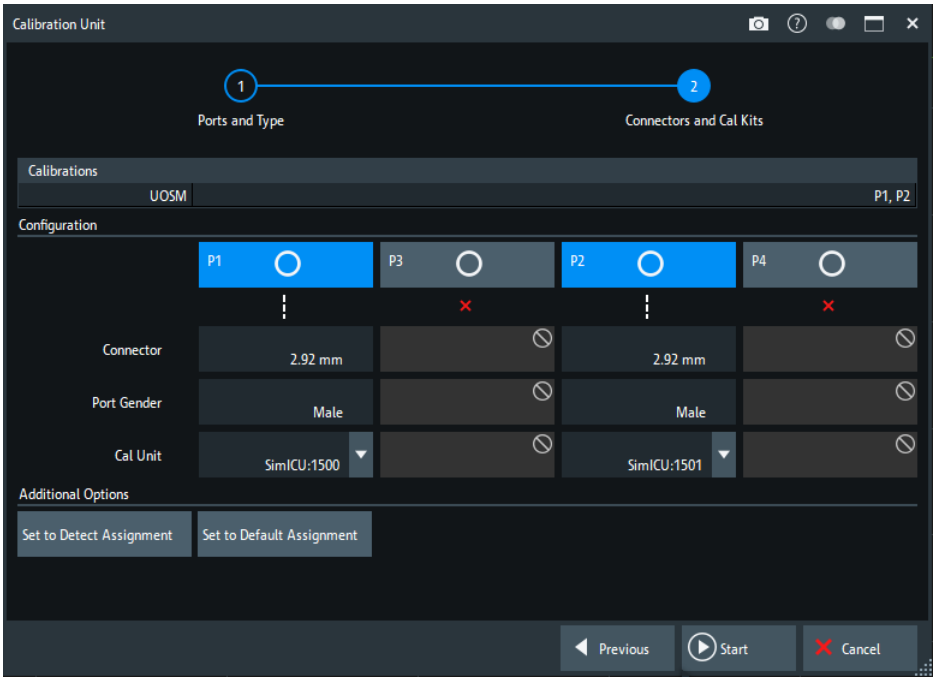


Figure 5-33: Calibration Unit wizard, step 2: Connections

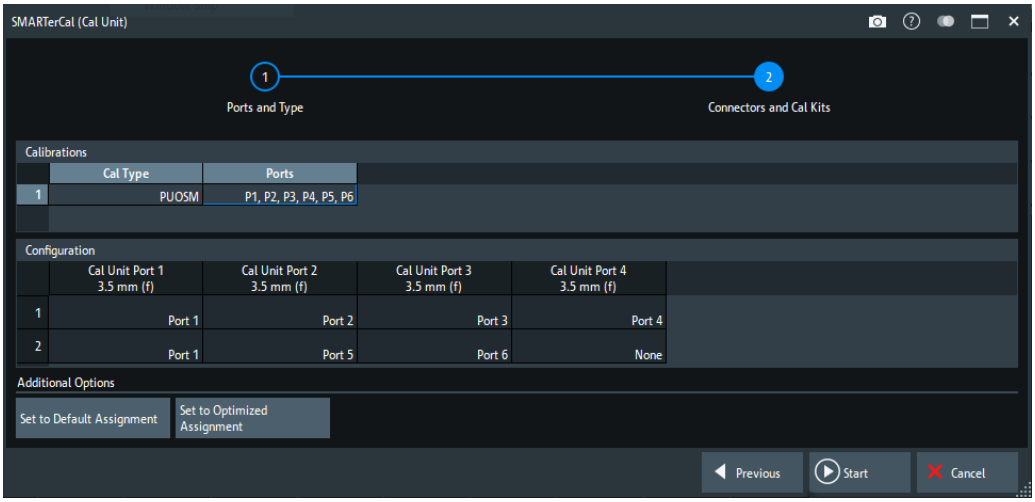


Figure 5-34: Calibration Unit wizard, step 2: Connections (>4 test ports and multiple port assignments)

If multiple port assignments are required and "Use Reduced Number of Throughs for Cal Unit" on page 814 is enabled, make sure that each port assignment contains the Common Port.

Optimized port assignment

If there is a switch matrix on the signal path, you can select between default and optimized port assignment (see "Optimized port assignment" on page 280)

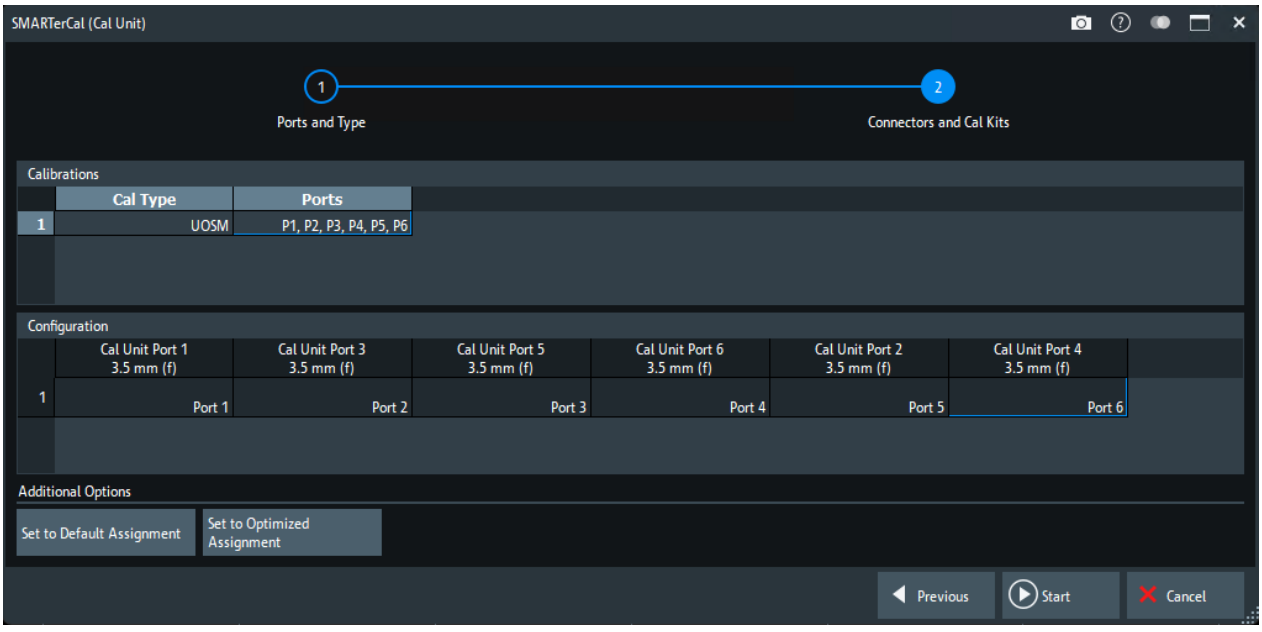


Figure 5-35: Optimized port assignment for 6-port R&S ZN-Z154

Port Assignment (manual)
The "Connections" step allows you to configure the assignments between VNA ports and calibration unit ports manually.

The R&S ZNB always proposes an optimum solution (minimum number of assignments) that also minimizes the physical port reconnections required between calibration stages. For user-modified assignments, it provides assistive information indicating insufficient, or redundant entries.

The test port connectors are automatically set according to the connector type of the selected calibration unit port.

Remote command:
`[SENSe<Ch>:]CORRection:COLLect:AUTO:ASSignment<Asg>:DEFine`
`[SENSe<Ch>:]CORRection:COLLect:AUTO:ASSignment<Asg>:DEFine:TPORt`
`[SENSe<Ch>:]CORRection:COLLect:AUTO:ASSignment:COUNt?`

Detect Assignment
Starts a procedure by which the R&S ZNB (with a little help from the attached calibration unit) auto-detects the connected ports. The automatic assignment replaces the configured one.

If auto-detection fails because of a high attenuation in the signal path, you can either enter the port assignment manually or connect matching port numbers and select "Default Port Assignment".

Remote command:
`[SENSe:]CORRection:COLLect:AUTO:PORTs:CONNection?`

Set to Default Port Assignment
Restores the default port assignments.

Remote command:
`[SENSe<Ch>:]CORRection:COLLect:AUTO:ASSignment:DEFine:DEFault`
`[SENSe<Ch>:]CORRection:COLLect:AUTO:ASSignment:DEFine:TPORt:DEFault`

Set to Optimized Assignment
Selects a port assignment that is optimized for measuring the passive load matches of the matrix test ports.

The optimized assignment is particularly recommended for calibration units with high through path losses, such as the R&S ZN-Z154. For other calibration units, usually it coincides with the default assignment, whose cabling can be a bit simpler.

If your measurement setup contains VNA ports that are not connected to a switch matrix, it is recommended to include (at least) one of them in the calibration. In the optimized port assignment, it is used as the source port for all passive load match measurements, and as the common port for all (unknown) Through measurements during calibration.

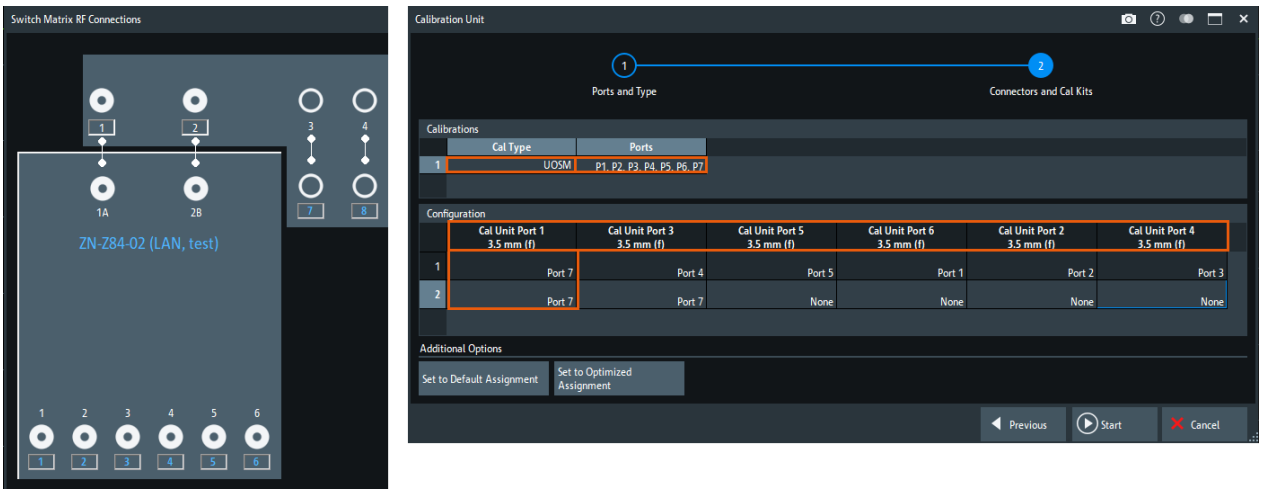


Figure 5-36: Direct VNA ports

Remote command:
`[SENSe<Ch>:]CORRection:COLLect:AUTO:ASSignment:DEFine:OPTimized`

Start
Proceeds to Step 3: Cal Unit.

If the configured port assignments are invalid, this action is disabled.

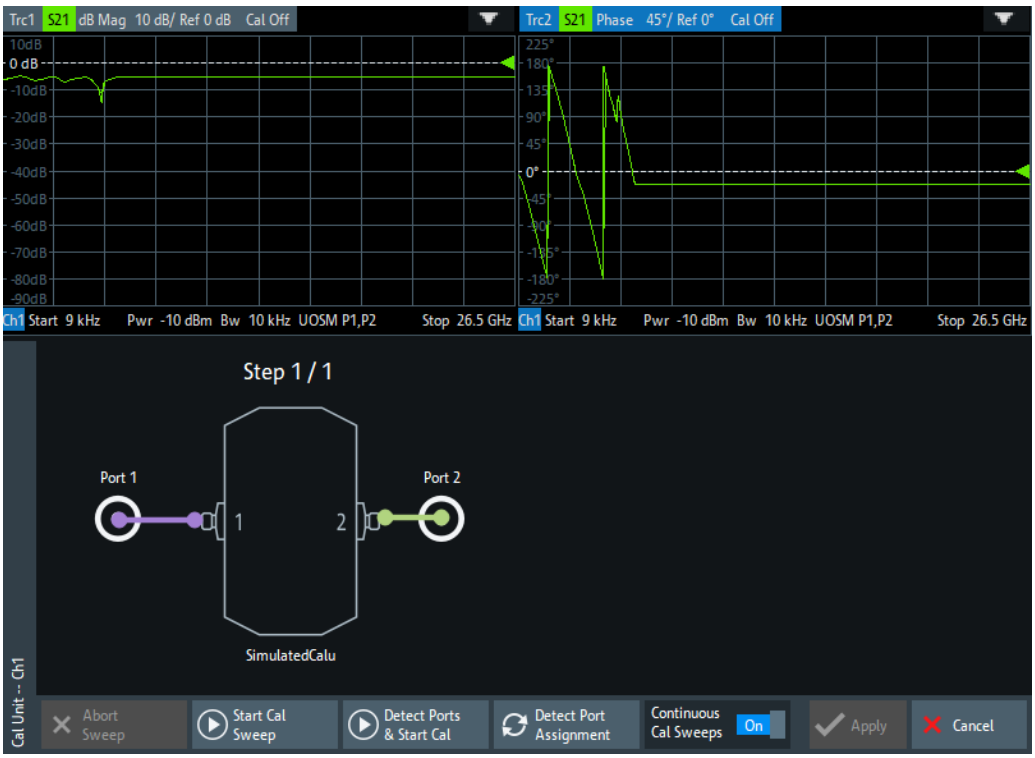
Step 3: Cal Unit

During the calibration phase, the R&S ZNB displays a "Cal Unit" screen that guides the user through the actual correction data acquisition.

In each data acquisition step:

- The calibration unit must be connected according to the depicted port assignment; auto-detection is possible
- The related test ports are calibrated with the "Calibration Type" selected in Step 1: Ports
- A subsweep is performed for each required test port or port pair and every required standard
  (and for every possible path between the port pairs, if external switch matrices are involved)

After all these calibration steps have been completed, the resulting system error correction can be calculated and applied.



In the upper part of the "Cal Unit" screen, the R&S ZNB shows the calibration sweep diagrams for the currently measured S-parameter. The lower part visualizes the active port assignment and the measurement progress.

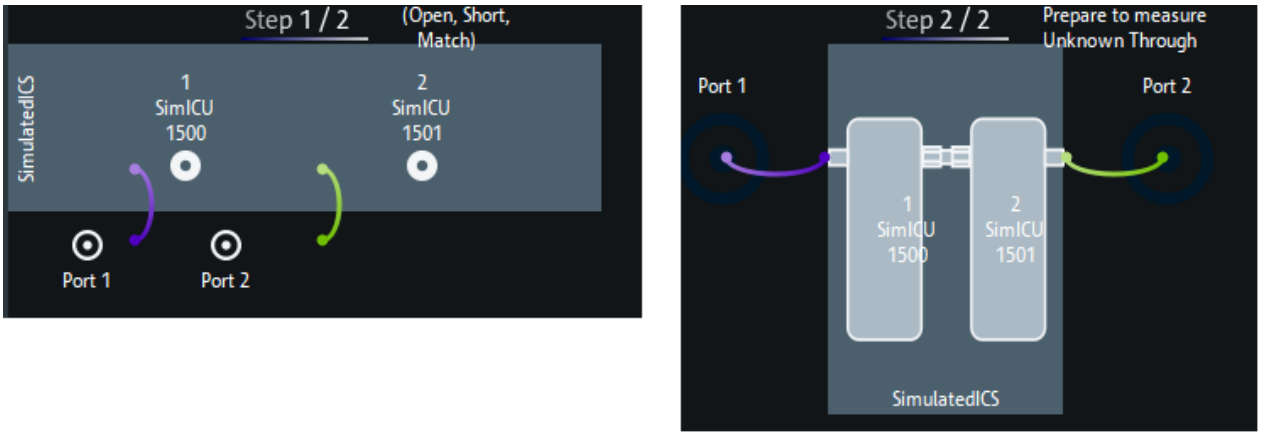
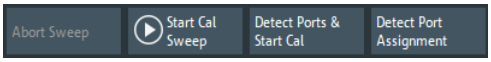


Figure 5-37: Automatic calibration: inline calibration system

If you are calibrating a matrix setup and have selected "Set to Optimized Assignment" in the previous calibration step, the detection functionality slightly changes. Because a different cabling can invalidate the optimization logic, it is not possible to adjust the assignment to the auto-detected one. In this scenario, you can check the port assignment and continue with the calibration only if the check was successful.



Calibration Sweep Diagrams
During the calibration sweep, each diagram contains a single S-parameter trace and a typical result trace for the measured calibration standard.

If switch matrices are involved, a sweep is performed for every possible signal path and for each of these paths a separate S-parameter trace is shown (see Section 4.7.30.5, "Multiport calibration", on page 277).

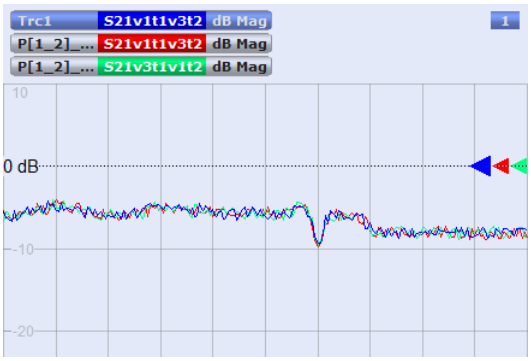


Figure 5-38: Multiple traces due to multiple paths

The purpose of the typical result traces "Trc1"" and "Trc2" is to avoid connection errors and to track hardware problems: if the correct standard type is measured, and everything is properly connected, then the measured traces are expected to be similar to the typical trace.

The S-Parameter traces are labeled P[*j*_*i*]_<standard type> S*ij*, where *j* indicates the input (test) port and *i* indicates the output port, e.g. P[1_2]_Unknown_Through S21.

If switch matrices are involved, the label also indicates:
- which VNA port *b* is switched to the input port *j* and
- which VNA port *a* is switched to the output port *i*

Instead of a single S*ij* trace we have multiple traces S*ij*v*a*t*i*v*b*t*j*. E.g., "S21v1t1v2t2" means that S21 is measured with VNA port 1 connected to the input port and VNA port 2 connected to the output port.

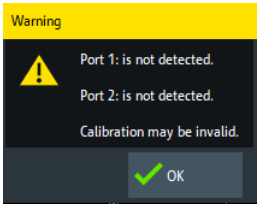
Start Cal Sweep / Abort Sweep
Starts the calibration sweep for the related port assignment or aborts it.

Remote command:
`[SENSe<Ch>:]CORRection:COLLect:AUTO:ASSignment<Asg>:ACQuire`

Detect Ports & Start Cal
Performs the "Detect Assignment" and "Start Cal Sweep" functions, one after the other.

Detect Assignment
Starts a procedure by which the R&S ZNB (with a little help from the attached calibration unit) auto-detects the connected ports. The automatic assignment replaces the configured one.

In case auto-detection fails
- an error report is shown as a warning dialog
- the undetected port connections are marked with warning signs
- the calibration can be invalid



Remote command:
[SENSe:]CORRection:COLLect:AUTO:PORTs:CONNection?

Check Ports & Start Cal / Check Port Assignment
Only available for matrix setups, and if you have selected "Set to Optimized Assignment" in the previous calibration step.

"Check Port Assignment" checks whether the detected port connections coincide with the optimized assignment. "Check Ports & Start Cal" proceeds with the calibration if no deviation is found.

Remote command:
[SENSe<Ch>:]CORRection:COLLect:AUTO:ASSignment<Asg>:CHECk?
[SENSe<Ch>:]CORRection:COLLect:AUTO:ASSignment<Asg>:ACQuire

Continuous Cal Sweeps
See "Continuous Cal Sweeps" on page 814.

Prev/Next
Navigates between the port assignments. Only available if multiple port assignments are required.

Apply/Cancel
Apply the calculated system error correction to the active channel (or to all channels in the active recall set, if all channels were calibrated).

The "Apply" button is enabled when calibration sweeps have been successfully performed for all required port assignments.

Remote command:
[SENSe<Ch>:]CORRection:COLLect:AUTO
[SENSe<Ch>:]CORRection:COLLect:AUTO:PORTs
[SENSe<Ch>:]CORRection:COLLect:AUTO:PORTs:TYPE
[SENSe<Ch>:]CORRection:COLLect:AUTO:TYPE
[SENSe<Ch>:]CORRection:COLLect:AUTO:SAVE
Extended cal unit settings:
MMEMory:AKAL:FACTory:CONVersion
MMEMory:AKAL:USER:CONVersion
SYSTem:COMMunicate:AKAL:CONNection
SYSTem:COMMunicate:AKAL:MMEMory[:STATe]

5.11.1.4 Calibration Unit wizard (MultiCal)

The "MultiCal" version of the Calibration Unit wizard guides you through the setup and simultaneous acquisition of multiple automatic system error corrections.

Access: Channel – [Cal] > "Start Cal" > "Calibration" – "Start (Cal Unit)"

The wizard proceeds through the following steps:

1. **"Ports":** Allows you to define the system error corrections (test ports, calibration unit, characterization, calibration type).
2. **"Connections":** Allows you to define the port assignments between the R&S ZNB and the calibration unit or units.
3. **"Cal Unit":** Allows you to acquire measurement data for the selected port assignments and the required calibration standards (provided by the calibration unit). Calculates the system error correction data (error terms) from the measurement data and applies the result to the active channel.

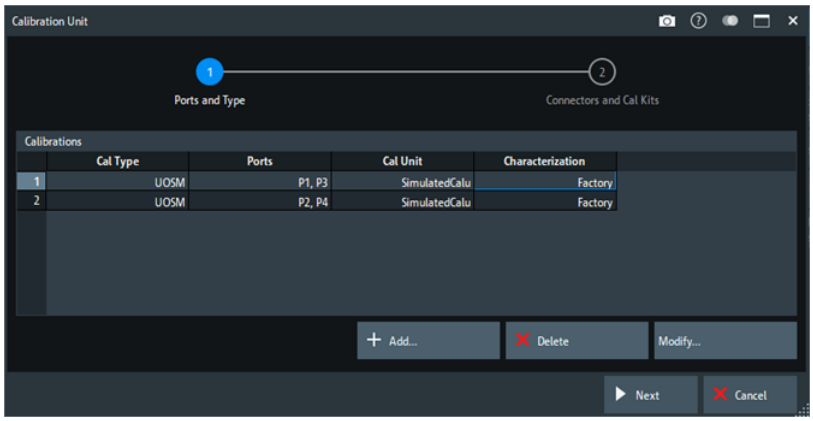
Background and related information

- The "Multiple Cal in Calibration Wizard" feature is disabled by default and has to be activated in the Calibration tab of the "System Config" dialog.
- Refer to Section 5.11.1.3, "Calibration Unit wizard", on page 565 for further information.

Step 1: Ports

The first page of the wizard presents the list of already configured calibrations and allows you to:

- "Add" new calibrations
- "Delete" or "Modify" existing calibrations



Add
Adds a new system error correction.

The "Add" button opens the Define Calibration dialog without pre-selected ports.

Delete
Deletes the selected calibration from the list of configured calibrations.

Modify
Edits the selected system error correction: opens the Define Calibration dialog with the corresponding ports and calibration type pre-selected.

Next
Proceeds to Step 2: Connections.

Inactive as long as no calibration is defined.

Define Calibration dialog

In MultiCal mode, the "Define Calibration" dialog allows you to define multiple system error corrections: for each system error correction, select the test ports to be calibrated and the calibration unit, characterization and calibration type to be used.

Access: Calibration Unit wizard (MultiCal), Step 1: "Ports" > "Add" / "Modify"

- The "Multiple Cal in Calibration Wizard" (MultiCal) feature is disabled by default and has to be activated in the Calibration tab of the "System Config" dialog.
- For background information on calibration types, see Section 4.5.1, "Calibration types", on page 155.

The layout of the main panel depends on the number of test ports available.

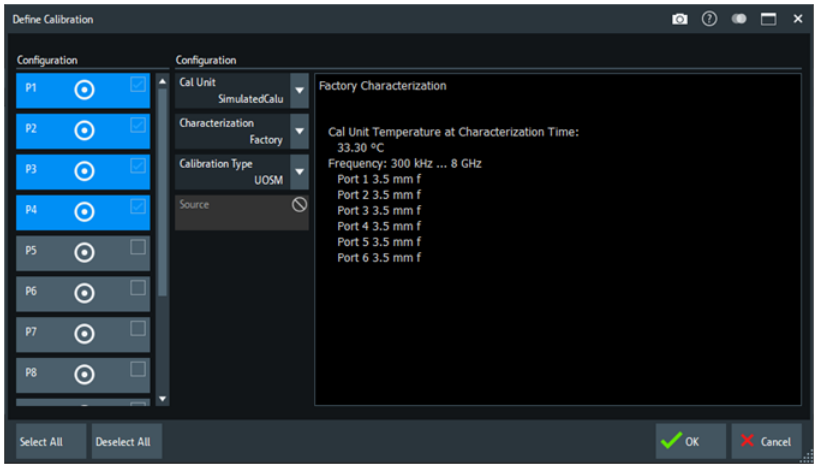


Figure 5-39: Define Calibration dialog (MultiCal, > 4 test ports)

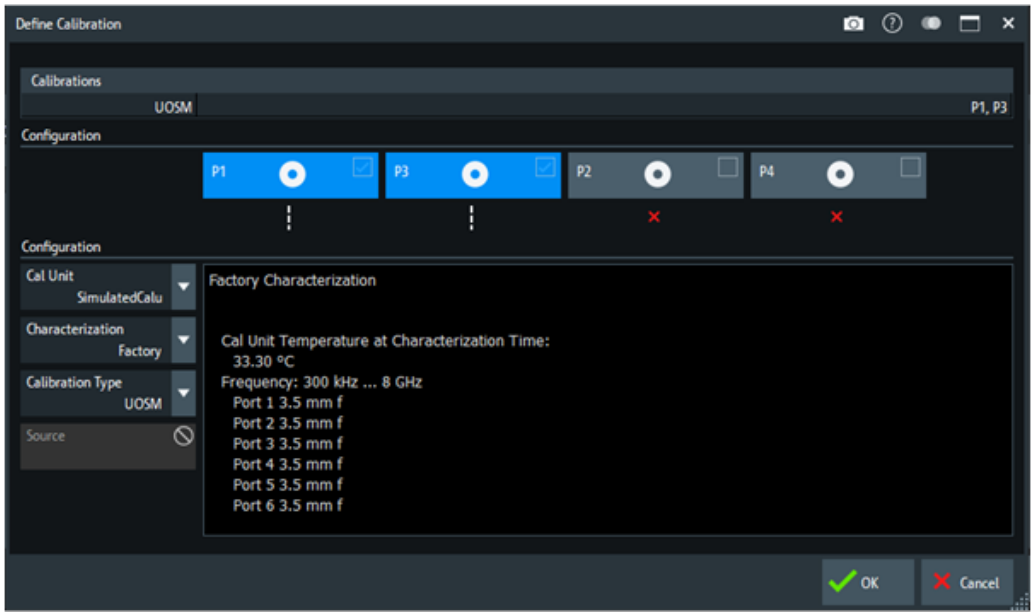


Figure 5-40: Define Calibration dialog (MultiCal, Inline Calibration System)

Ports
Selects the test ports to be calibrated.

Note: Calibration and port de-/activation.
The analyzer firmware automatically activates/deactivates ports during/after a (successful) calibration:

- Calibrated ports that were previously disabled, are automatically enabled as single-ended logical ports.
- An uncalibrated port that is not used by a measurement (i.e. the port is not required by any trace of the related channel) is disabled.

Remote command:
The port parameters in many calibration commands define the calibrated ports.

Cal Unit
Displays the connected calibration units. The R&S ZNB auto-detects all calibration units which are connected to one of its USB ports. If several cal units are connected, one of them must be selected for calibration (active cal unit). A warning is displayed if the current sweep range of the network analyzer exceeds the characterized frequency range of the calibration unit.

For background information, see Section 4.5.5, "Automatic calibration", on page 175.

Remote command:
`SYSTem:COMMunicate:RDEVice:AKAL:ADDRess:ALL?`
`SYSTem:COMMunicate:RDEVice:AKAL:ADDRess`

Characterization
See "Characterization" on page 568.

Temperature Compensation
Available for inline calibration systems (ICS) only. Applies to all inline calibration units (ICUs) that are connected to the ICS and that are used with factory characterization data.

If enabled, the R&S ZNB firmware adjusts the factory characterization data according to the ICU temperatures.

Remote command:
`[SENSe<Ch>:]CORRection:COLLect:AUTO:TEMPcomp`

Calibration Type / Source
Selects the calibration type for the selected physical ports. For an overview, refer to Table 4-7.

The reflection calibration types can be used for any combination of physical ports: reflection calibrations are performed for each selected port.

A transmission calibration type requires at least two physical ports. For the **unidirectional** transmission calibration types ("Trans Norm", "One Path Two Ports"), the direction (**"Source"** port) must be specified in addition.

Transmission normalization and "One Path Two Ports" calibration types require two-port (Through) characterization data for the cal unit. For current calibration units, however, these two-port characterizations are typically not part of the factory characterization. If a tooltip indicates missing two-port characterization data, simply perform a new characterization of your cal unit. In the first dialog of the "Characterization" wizard, select "Take All OSM and Through" to make sure that the necessary two-port data is acquired. See also "Characterization wizard" on page 645.

Remote command:
`[SENSe<Ch>:]CORRection:COLLect:AUTO:TYPE`
`[SENSe<Ch>:]CORRection:COLLect:AUTO:PORTs:TYPE`
`[SENSe<Ch>:]CORRection:COLLect:AUTO:CONFigure`

Step 2: Connections

Defines the port assignments between the R&S ZNB and the calibration unit.

Background information
Refer to Section 4.5.5.6, "Multiple port assignments", on page 185 for details on multiple port assignments.

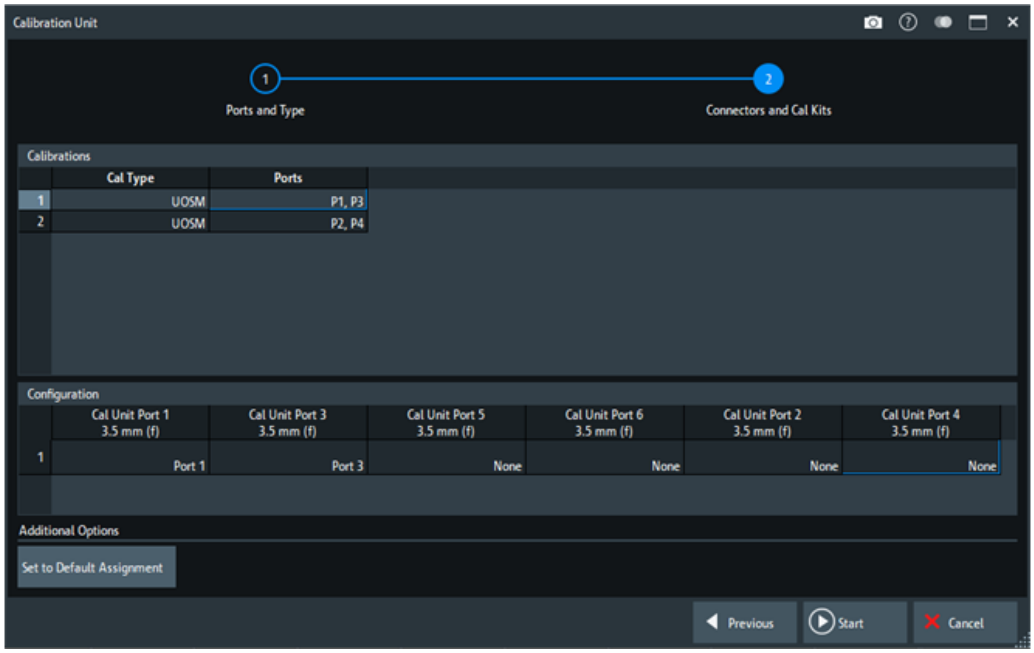


Figure 5-41: Multiple calibrations (and multiple port assignments)

Similar functionality as for the "SingleCal" version of the wizard (see "Step 2: Connections" on page 570): select the adequate calibration in the "Cal Type"/"Ports" table to display and edit the corresponding port assignments.

Back
Go back to Step 1: Ports.

Start
Proceeds to Step 3: Cal Unit.

If the configured port assignments are invalid, this action is disabled.

Step 3: Cal Unit

During the calibration phase, the R&S ZNB displays a "Cal Unit" screen that guides the user through the correction data acquisition.

Same logic as in the single calibration version of the wizard (see "Step 3: Cal Unit" on page 573).

5.11.1.5 Calibration Setting wizard

The "Calibration Setting" wizard guides you through the setup and execution of a manual system error correction.

Access: Channel – [Cal] > "Start Cal" > Calibration – "Start (Manual)"

The wizard proceeds through the following steps:

1. **"Ports and Type":** Select the ports you want to calibrate and the calibration type you want to use.
2. **"Connectors and Cal Kits":** For all ports you want to calibrate, select the connector type, gender and cal kit. If necessary, import a calibration kit.
3. **"Calibration":** Acquire measurement data for the required ports or port pairs and the required standards. Finally, decide whether to apply the resulting system error correction or to discard it.

Background and related information

- Refer to Section 4.5, "Calibration", on page 154 for background information
- Refer to Section 5.11.1.6, "Calibration Setting wizard (MultiCal)", on page 591 for the "MultiCal" variant of the wizard
- If the active channel is already equipped with a system error correction, the "Calibration Setting" wizard loads the underlying setup. If the calibration setup is not changed and sweep data are available from previous calibrations, the existing system error correction can be optimized without repeating the measurement of all standards.
- When you apply the new system error correction, the current calibration is replaced and discarded.
  To persist the current calibration, you can transfer it to the calibration "Pool" using the Calibration Manager dialog.
- The active system error correction data can be read (and modified) using the remote control command `[SENSe<Ch>:]CORRection:CDATa`.

Step 1: Ports and Type

Allows you to select the test ports to be calibrated and the calibration type to be used.

Background information

Refer to Section 4.5.1, "Calibration types", on page 155.

The layout of the main panel depends on the number of test ports available:

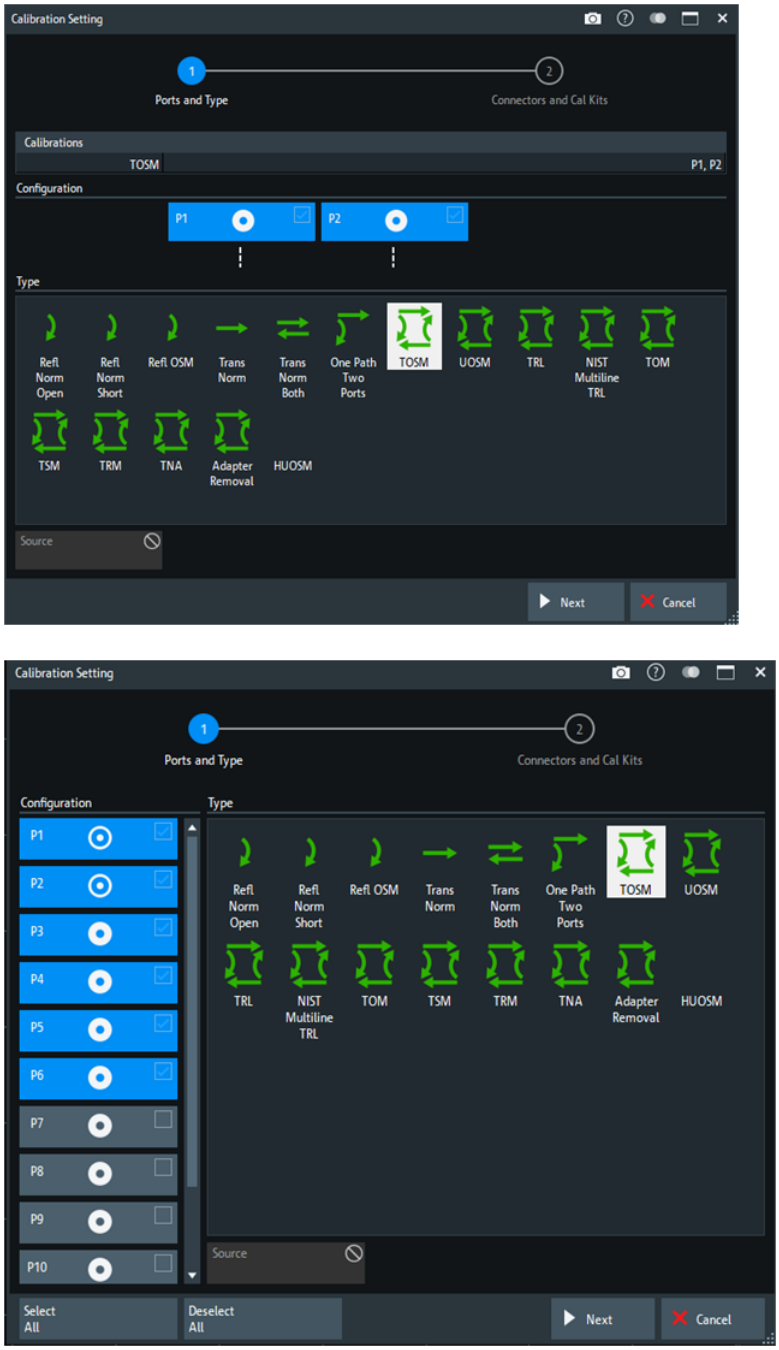


Figure 5-42: Calibration Presetting: Ports and Type (> 4 test ports)

Ports
Selects the test ports to be calibrated.

Note: Calibration and port de-/activation.
The analyzer firmware automatically activates/deactivates ports during/after a (successful) calibration:

- Calibrated ports that were previously disabled, are automatically enabled as single-ended logical ports.

- An uncalibrated port that is not used by a measurement (i.e. the port is not required by any trace of the related channel) is disabled.

Remote command:
The port parameters in many calibration commands define the calibrated ports.

Type/Source
Selects the calibration type. The green arrow symbols give a preview of the type and the number of calibration sweeps involved:

- Curved arrows (example: "Refl Norm Open") denote one or more reflection measurements at each port.
  Reflection calibration types can be used for any set of test ports: reflection calibrations are repeated for each port.
- Straight, horizontal arrows (example: "Trans Norm") denote one or more transmission measurements between each pair of two ports.
  Transmission calibration types require at least two physical ports. For unidirectional transmission calibration types ("Trans Norm", "One Path Two Ports"), the (**"Source"** port) must be specified in addition.
- The full n-port calibration types (n > 1, e.g. "TOSM") are symbolized by a closed square symbol. The number of arrows increases the complexity but can also improve the accuracy of the calibration.

For an overview, refer to Table 4-7.

Remote command:
`[SENSe<Ch>:]CORRection:COLLect:METHod:DEFine`

Calibrate all Channels
Enable this control to apply the acquired correction data to all channels in the active recall set. Leave it disabled (preset setting) to apply them only to the active channel.

Note that this option is available only if the active recall set contains multiple channels.

Remote command:
`[SENSe:]CORRection:COLLect:CHANnels:ALL`

Select All/Deselect All
Selects/deselects all ports.

These buttons are only shown if more than 4 ports are available.

Next
Proceeds to Step 2: Connectors and Cal Kits.

Step 2: Connectors and Cal Kits

Selects the connector type and gender for all ports and allows you to import a calibration kit.

Background information
Refer to Section 4.5.2, "Calibration standards and calibration kits", on page 168.

Messages in the dialog

An information message (or error message) is displayed if one of the following happens:

- One of the selected calibration kits is described by ideal kit parameters or typical values.
- One of the selected calibration kits does not contain all standards that are required for the previously selected calibration type.
- Different connector types are defined at the ports but the selected calibration type requires uniform connectors.
- A cal kit standard does not cover the entire calibrated frequency range.

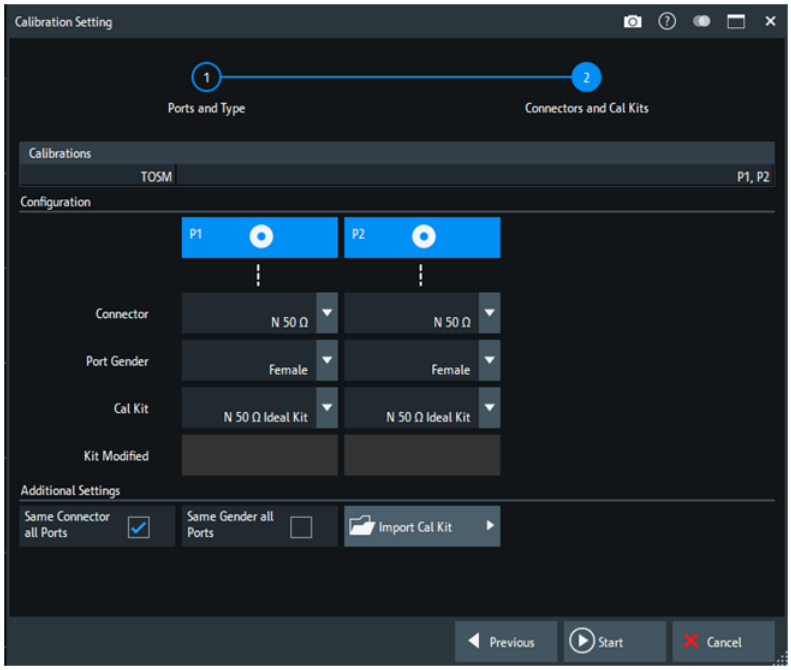


Figure 5-43: Calibration Setting Wizard, Step 2: Connectors and Cal Kits

The upper part of the panel shows the ports and the calibration type selected in Step 1: Ports and Type. The lower part gives access to the connector and cal kit settings.

If more than 4 ports are available, the GUI is slightly different:

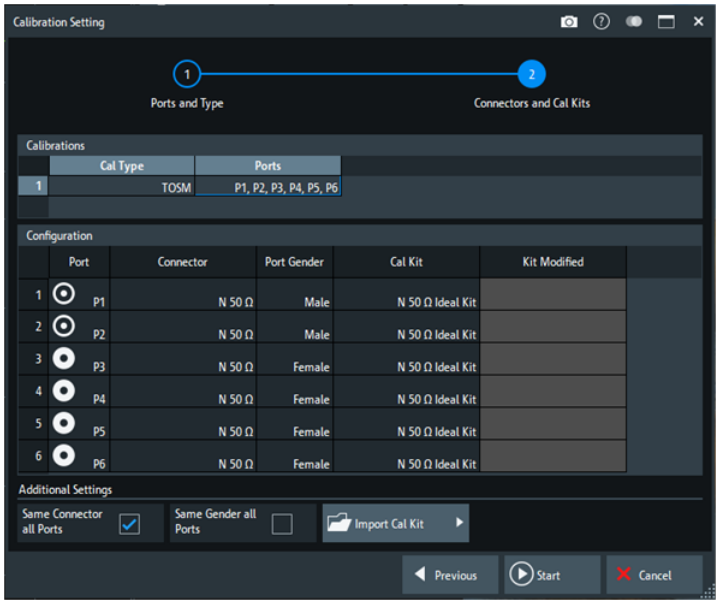


Figure 5-44: Calibration Setting: Connectors and Cal Kits (>4 ports)

Connector / Port Gender
Defines the connector types and genders of the ports to be calibrated. For symmetric (sexless) connector types (e.g. 7 mm / PC7), "Port Gender" is unavailable.

If "Same Connector All Ports" is active, the connector types at all ports (but not their gender) are always adjusted to the current selection. If "Same Gender All Ports" is active, the genders at all ports are always adjusted to the current selection.

User-defined connectors can be added or removed in the Cal Connector Types dialog.

Remote command:
`[SENSe<Ch>:]CORRection:COLLect:CONNection<PhyPt>`
`[SENSe<Ch>:]CORRection:COLLect:SCONnection<PhyPt>`
`[SENSe:]CORRection:CONNection`
`[SENSe:]CORRection:CONNection:CATalog?`
`[SENSe<Ch>:]CORRection:CONNection:DELete`

Cal Kit
Selects a cal kit for the connector at each selected physical port. The drop-down list contains all available calibration kits for the selected connector type.

The assignment of a calibration kit to a connector type must be the same for all physical ports: If a calibration kit is changed, the R&S ZNB automatically assigns the new kit to all ports with the same connector type.

Use "Import Cal Kit" to add new kits to the list.

Remote command:
`[SENSe:]CORRection:CKIT:SELect`

Same Connector All Ports / Same Gender All Ports
Assigns the same connector type or gender to all selected physical ports. For some multi-port calibration types, the port connector types must be equal, e.g. because they require a Through standard with known characteristics.

Remote command:
`[SENSe<Ch>:]CORRection:COLLect:CONNection:PORTs`
`[SENSe<Ch>:]CORRection:COLLect:CONNection:GENDers`

Import Cal Kit
Opens the "Import Calibration Kit" dialog that allows you to import a cal kit file. For background information, see Section 4.5.2.4, "Cal kit Files", on page 173.

Remote command:
`MMEMory:LOAD:CKIT`

Back
Go back to Step 1: Ports and Type.

Start
Start Step 3: Calibration.

Step 3: Calibration

Allows you to acquire error correction data for every required port (pair) and calibration standard, where "required" depends on the selected ports and calibration type. On "Apply", the R&S ZNB calculates the system error correction (error terms) from the measurement data of the standards and applies the result to the active channel.



The "Reduced Through" logic helps to keep the number of transmission measurements as small as possible (see Section 4.5.1.11, "Full n-Port calibration with reduced number of Through connections", on page 166).



Figure 5-45: Calibration Setting Wizard, Step 3: Calibration

In the upper part of the "Calibration" screen, the R&S ZNB shows the sweep diagrams for the currently measured S-parameter. The lower part displays the calibrated ports and standards and visualizes the measurement progress.

Calibration Sweep Diagrams

During the calibration sweep, each diagram contains a single S-parameter trace and a typical result trace for the measured calibration standard.

If switch matrices are involved, a sweep is performed for every possible signal path and for each of these paths a separate S-parameter trace is shown (see Section 4.7.30.5, "Multiport calibration", on page 277).

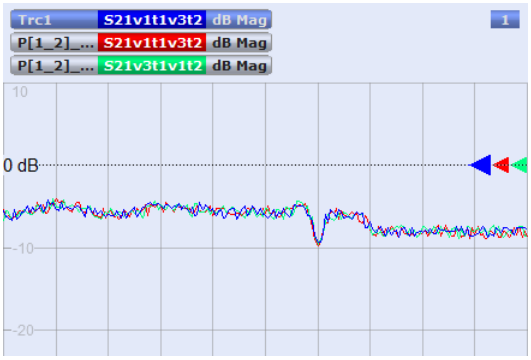


Figure 5-46: Multiple traces due to multiple paths

The purpose of the typical result traces "Trc1"" and "Trc2" is to avoid connection errors and to track hardware problems: if the correct standard type is measured, and everything is properly connected, then the measured traces are expected to be similar to the typical trace.

The S-Parameter traces are labeled P[*j*_*i*]_<standard type> S*ij*, where *j* indicates the input (test) port and *i* indicates the output port, e.g. P[1_2]_Unknown_Through S21.

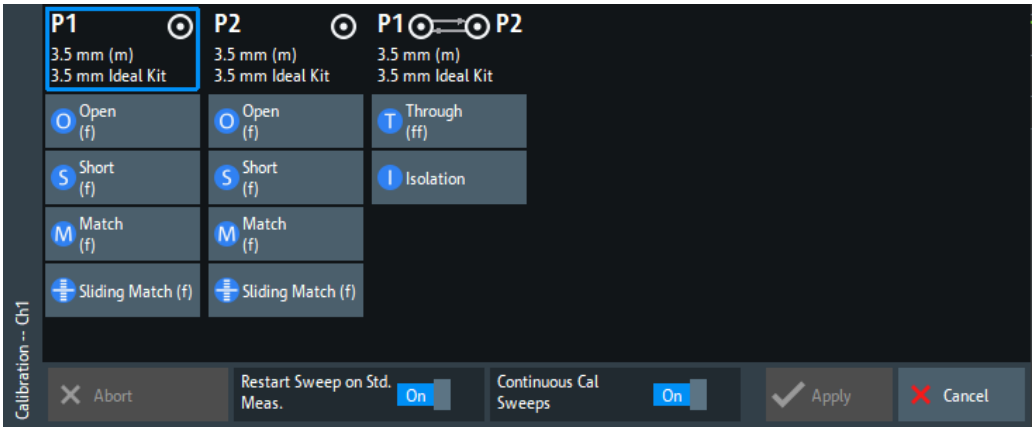
If switch matrices are involved, the label also indicates:

- which VNA port *b* is switched to the input port *j* and
- which VNA port *a* is switched to the output port *i*

Instead of a single S*ij* trace we have multiple traces S*ij*va*i*vb*tj*. E.g., "S21v1t1v2t2" means that S21 is measured with VNA port 1 connected to the input port and VNA port 2 connected to the output port.

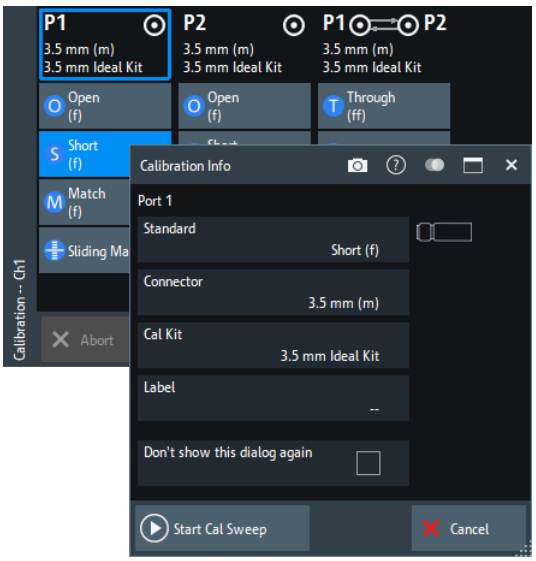
Start Cal Sweep

The dock widget below the diagrams shows the calibrated ports and standards and visualizes the measurement progress.



Use the buttons representing the calibration standards to start the corresponding calibration sweeps.

If "Show Cal Kit Label" is enabled on the Calibration tab of the "System Config" dialog, an additional "Calibration Info" dialog is displayed. In this case, the cal sweep is started from this dialog.



"Don't Show this Dialog Again" has the same effect as disabling "Show Cal Kit Label".

A green checkmark indicates that the calibration data of a standard has been acquired successfully. A green checkmark after the port symbol indicates that the minimum number of calibration measurements for the port has been performed.

Tip:

- For a TRL calibration, at least one line standard must be measured between any pair of calibrated ports. See Section 4.5.1.9, "TRL calibration", on page 163.
- If the selected calibration kit comprises a Sliding Match, then for every required Match measurement either the Match or at least three positions of the Sliding Match must be measured. See Section 4.5.2.3, "Sliding Match standards", on page 172.

Remote command:
`[SENSe<Ch>:]CORRection:COLLect[:ACQuire]:SELected`
See also: `[SENSe<Ch>:]CORRection:COLLect:LOAD:SELected`

Restart Sweep on Std. Meas.
If this function is active, a new standard measurement initiates a new sweep, starting at the beginning ("Start") of the sweep range: The sweep points for the calibration sweep are in ascending order, like for an ordinary measurement.

If "Restart Sweep on Std. Meas." is inactive, the new standard measurement is started at the current sweep point; the current sweep is continued as a calibration sweep.

Note:
This function has a secondary effect in IDLE mode (i.e. while the calibration sweep is NOT running):

- If active, only the selected trace is refreshed.
- If inactive, the R&S ZNB permanently refreshes all traces of all diagrams which can put a heavy load on the connected switch matrices (if any).

Hence it is recommended (and default) to activate it, in particular if one of the matrices uses *mechanical* switches (that wear off apart from making noise).

Continuous Cal Sweeps
See "Continuous Cal Sweeps" on page 814.

Apply
Is enabled when sufficient data have been acquired for the calibrated ports and standards. The button starts the calculation of the system error correction and closes the calibration wizard. The current instrument settings are stored with the correction data.

To avoid incompatibilities, older calibration data is deleted unless it has been transferred into the calibration "Pool" using the "Calibration Manager" (see Section 5.11.4.3, "Calibration Manager dialog", on page 657).

Note: Checks during the calculation of correction data
Incompatibilities between the selected calibration type, the standards and the channel settings can cause the calibration to be inaccurate. The analyzer auto-detects potential sources of errors and displays appropriate, self-explanatory notice boxes.

Remote command:
```
[SENSe<Ch>:]CORRection:COLLect:SAVE:SELected[:DUMMy]
[SENSe<Ch>:]CORRection:COLLect:SAVE:SELected:DEFault
[SENSe<Ch>:]CORRection:COLLect:DELete
[SENSe<Ch>:]CORRection:DATA:PARameter<Sfk>?
[SENSe<Ch>:]CORRection:DATA:PARameter<Sfk>:PORT<PhyPt>?
[SENSe<Ch>:]CORRection:DATE?
[SENSe<Chn>:]CORRection:SSTate?
[SENSe<Ch>:]CORRection:STIMulus?
[SENSe<Ch>:]CORRection:STIMulus:PORT<PhyPt>?
```

5.11.1.6 Calibration Setting wizard (MultiCal)

The "MultiCal" version of the Calibration Setting wizard guides you through the setup and simultaneous manual acquisition of multiple system error corrections.

Access: Channel – [Cal] > "Start Cal" > "Start (Manual)"

The wizard proceeds through the following steps:

1. **"Ports and Type":** Allows you to define the system error corrections you want to perform (test port sets, calibration types).
2. **"Connectors and Cal Kits":** Allows you to define the connector type and gender of the calibrated ports and to select the appropriate cal kits. If necessary, you can import a calibration kit.
3. **"Calibration":** Allows you to acquire measurement data for the required ports or port pairs and the required calibration standards. Calculates the system error correction data (error terms) from the measurement data and applies the result to the active channel.

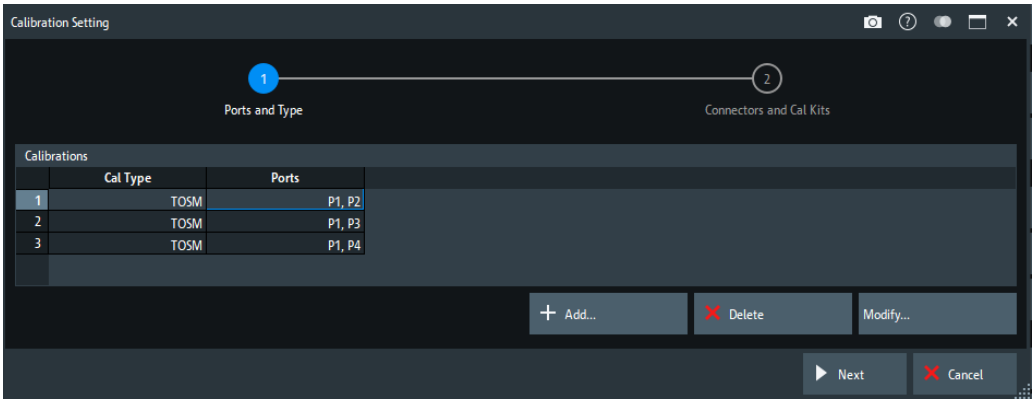
Background and related information

- The "Multiple Cal in Calibration Wizard" feature is disabled by default and has to be activated in the Calibration tab of the "System Config" dialog.
- Refer to Section 5.11.1.5, "Calibration Setting wizard", on page 581 for further information.

Step 1: Ports and Type

The first page of the wizard presents the list of already configured calibrations and allows you to:

- "Add"new calibrations
- "Delete" or "Modify" existing calibrations



Add
Adds a new system error correction.

The "Add" button opens the Define Calibration dialog without pre-selected ports.

Delete
Deletes the selected calibration from the list of configured calibrations.

Modify
Edits the selected system error correction: opens the Define Calibration dialog with the corresponding ports and calibration type pre-selected.

Next
Proceeds to Step 2: Connectors and Cal Kits.

Inactive as long as no calibration is defined.

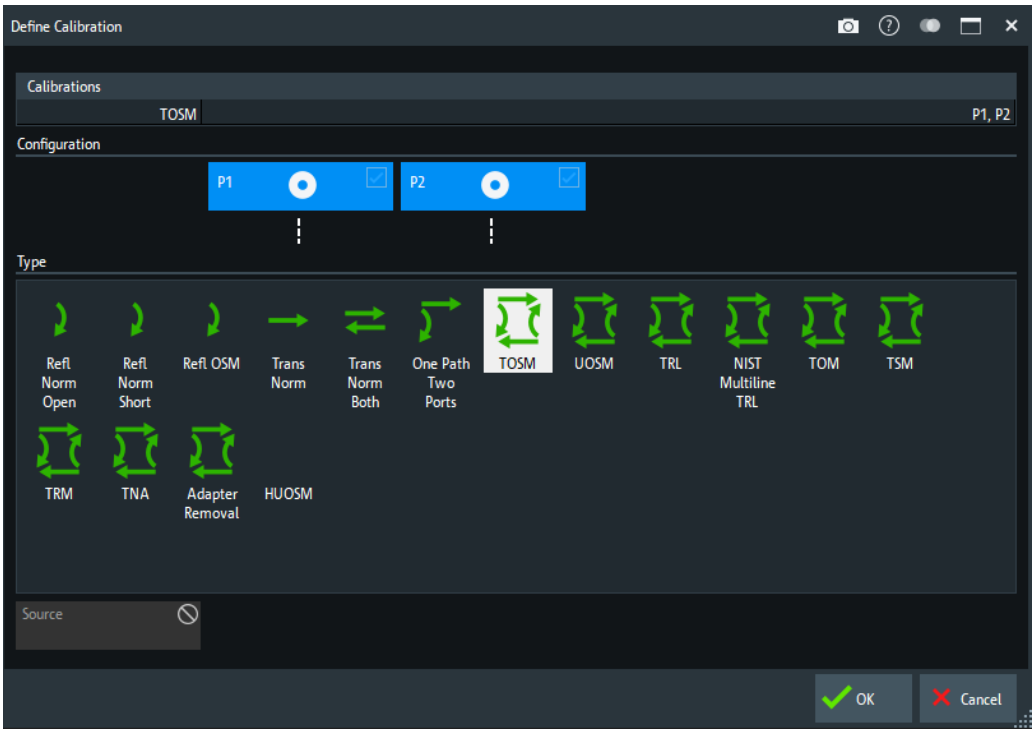
Define Calibration dialog

you to select the test ports to be calibrated and the calibration type to be used.

Background information

Refer to Section 4.5.1, "Calibration types", on page 155 for background information.

Access: Calibration Setting wizard (MultiCal) > Step 1: Ports and Type> "Add" / "Modify"



The layout of the dialog panel depends on the number of test ports available.



Figure 5-47: Define Calibration dialog (manual, > 4 test ports)

Ports
Selects the test ports to be calibrated.

Note: Calibration and port de-/activation.
The analyzer firmware automatically activates/deactivates ports during/after a (successful) calibration:

- Calibrated ports that were previously disabled, are automatically enabled as single-ended logical ports.
- An uncalibrated port that is not used by a measurement (i.e. the port is not required by any trace of the related channel) is disabled.

Remote command:
The port parameters in many calibration commands define the calibrated ports.

Type/Source
Selects the calibration type. The green arrow symbols give a preview of the type and the number of calibration sweeps involved:

- Curved arrows (example: "Refl Norm Open") denote one or more reflection measurements at each port.
  Reflection calibration types can be used for any set of test ports: reflection calibrations are repeated for each port.
- Straight, horizontal arrows (example: "Trans Norm") denote one or more transmission measurements between each pair of two ports.
  Transmission calibration types require at least two physical ports. For unidirectional transmission calibration types ("Trans Norm", "One Path Two Ports"), the (**"Source"** port) must be specified in addition.
- The full n-port calibration types (n > 1, e.g. "TOSM") are symbolized by a closed square symbol. The number of arrows increases the complexity but can also improve the accuracy of the calibration.

For an overview, refer to Table 4-7.

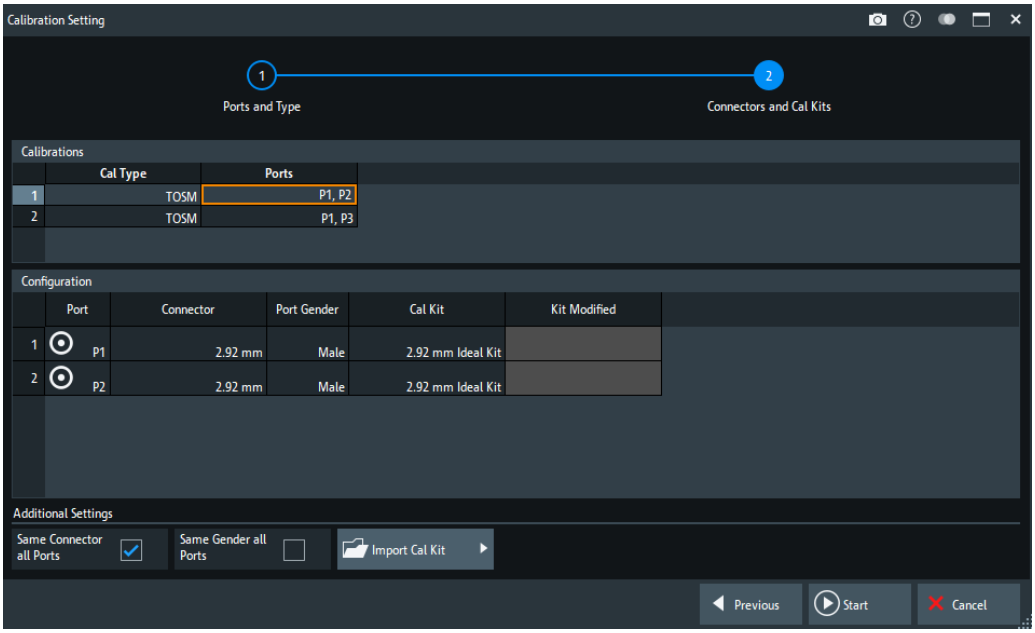
Remote command:
`[SENSe<Ch>:]CORRection:COLLect:METHod:DEFine`

Select All/Deselect All
Selects/deselects all ports.

These buttons are only shown if more than 4 ports are available.

Step 2: Connectors and Cal Kits

Selects the connector type, gender and cal kit for all ports and allows you to import a calibration kit.



The upper part of the panel shows the previously configured calibrations. Select the appropriate one to get access to the related ports. For the individual calibrations, the functionality is the same as described for the "SingleCal" version of the wizard (see "Step 2: Connectors and Cal Kits" on page 584).

Back
Go back to Step 1: Ports and Type.

Start
Start Step 3: Calibration.

Step 3: Calibration

Allows you to acquire error correction data for every required port (pair) and calibration standard, where "required" depends on the selected ports and calibration types.

Same logic as in the single calibration version of the wizard (see "Step 3: Calibration" on page 587).

The possibility to "share" measurements between the calibrations and the "Reduced Through" logic help to keep the number of measurements as small as possible (see Section 4.5.1.11, "Full n-Port calibration with reduced number of Through connections", on page 166).

5.11.1.7 Power Cal wizard

The "Power Cal" wizard displays a single screen, showing the current source and receive ports. It allows you to perform "Ref. Receiver", "Meas. Receiver" and scalar source power ("Source Flatness") calibrations, based on the current power calibration settings (see Section 5.11.3, "Power Cal Settings tab", on page 648).

Access: Channel – [Cal] > "Start Cal" > "Power Cal"

Screen elements



From top to bottom, the screen consists of the following elements.

Calibration Sweep Diagram
The calibration sweep diagram in the upper part of the screen shows the progress of the calibration and the accuracy of a completed calibration ("Verification"). The diagram is scaled in "dB Mag" format.

The diagram title indicates the ongoing calibration type and reading. The traces in the diagram vary according to the calibration stage.

While no calibration is performed, or during a source power calibration ("Power"), the following traces are displayed:

- A limit line (double horizontal) represents the target power of the source power calibration ("Cal Power").
- "Pmtr<n>" shows the reading of the power meter <n> in use. This trace is only shown during the first calibration sweeps; the following sweeps are based on the reference receiver result.
- "a<m>(P<m>)" shows the source power reading of the analyzer (wave quantity, reference receiver) at the calibrated source port P<m>.

After a successful power calibration, the trace "a<m>(P<m>)" must be close to the "Cal Power".

During a measurement receiver calibration ("Meas. Receiver"), the following traces are displayed:

- The trace "a<m>(P<m>)" shows the (previously calibrated) power at the calibrated reference plane (source port P<m>).
- The trace "b<n>(P<m>)" shows the current power reading of the analyzer at the calibrated receive port P<n> (source port P<m>).

After successful measurement receiver calibration, the "b<n>(P<m>)" trace must be close to the "a<m>(P<m>)" trace. Due to the previous power calibration, both traces must be close to the cal power.

Remote command:

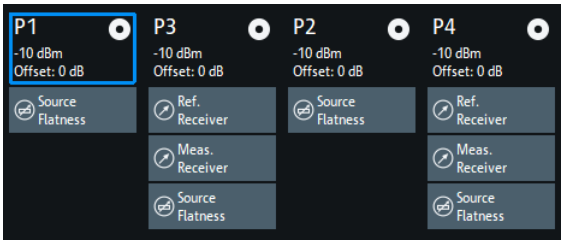
```
SOURce<Ch>:POWer<PhyPt>:CORRection[:ACQuire]:VERification:
RESult?
```

Port Overview

Shows all source ports together with the possible power calibrations.

For each analyzer port P1 ... PN, a "Ref. Receiver", a "Meas. Receiver" and a "Source Flatness" calibration can be performed.

- If a "SMARTerCal" is active, additional receiver power calibrations are not allowed and the R&S ZNB only offers the possibility to perform "Source Flatness" calibrations of the related test ports:



  See Section 4.5.7.3, "Combining SMARTerCal with scalar power calibration", on page 195 for details.
- If external generators are configured, they appear as additional source ports G1 ... Gm in the port overview. A source power (flatness) calibration is available at each generator port.

Select a "Ref. Receiver", "Meas. Receiver", or "Source Flatness" symbol to start the related calibration. See "Power Cal dialog – Ref. Receiver" on page 598, and "Power Cal dialog – Meas. Receiver" on page 599, and "Power Cal dialog – Source Flatness" on page 600.

A green checkmark ✓ indicates that the calibration data has been acquired successfully. A half circle overlay indicates that only the respective port frequencies are covered (see "Calibrate Only Port Frequency" on page 599).

Remote command:

```
SOURce<Ch>:POWer:CORRection[:ACQuire]
[SENSe<Ch>:]CORRection:POWer<PhyPt>:ACQuire
SOURce<Ch>:POWer:CORRection:DATA:PARameter<Wv>?
SOURce<Ch>:POWer<PhyPt>:CORRection:DATA:PARameter<Wv>:COUNt?
```

Apply
Is enabled when a new set of power calibration data has been acquired. The button applies all available power calibrations to the active channel, aborts the verification sweeps, and closes the port overview section.

The power calibration state is indicated in the trace list, see Section 4.5.6.4, "Power calibration labels", on page 191. Use the functions in the Section 5.11.4, "Use Cal tab", on page 655 to activate, deactivate, or store power calibrations.

Remote command:
```
SOURce<Ch>:POWer:CORRection[:ACQuire]
[SENSe<Ch>:]CORRection:POWer<PhyPt>:ACQuire
SOURce<Ch>:POWer:CORRection:DATA
SOURce<Ch>:POWer:CORRection:DATA:PORT<PhyPt>
[SENSe<Chn>:]CORRection:PSTate?
```

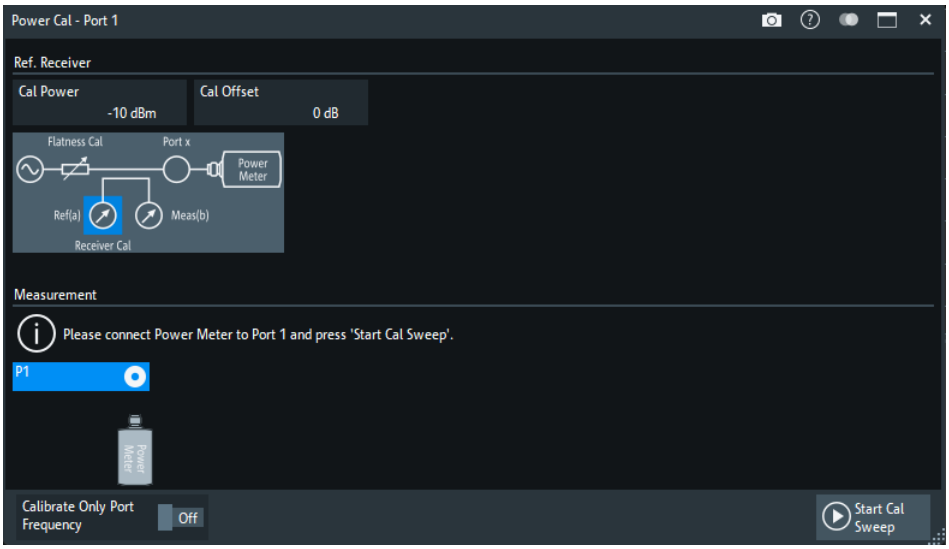
Power Cal dialog – Ref. Receiver

In Ref. Receiver mode, the "Power Cal" dialog guides you through a reference receiver calibration.

The selected source port is displayed with its cal power settings (see Section 5.11.3.2, "Cal Power Config dialog", on page 650). A circuit diagram visualizes the connectivity. A power meter must be connected to the calibrated source port.

Access: Power Cal wizard > Ref. Receiver

Background information
Refer to Section 4.5.6.2, "Source flatness calibration", on page 188.



Start Cal Sweep
Starts the calibration sweeps for the selected port and power calibration settings and closes the dialog. The calibration is performed as described in "Calibration procedure" on page 189.

Open the Power Cal Settings tab if you wish to modify the calibration procedure.

Remote command:
SOURce<Ch>:POWer:CORRection[:ACQuire]
SOURce<Ch>:POWer<PhyPt>:CORRection:COLLect[:ACQuire]

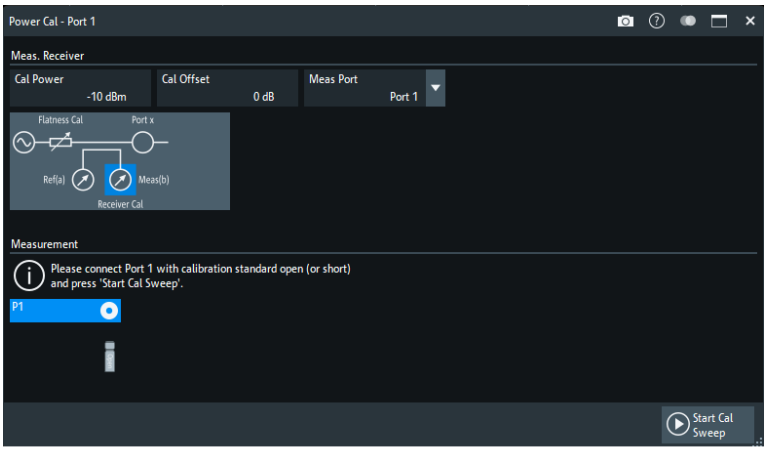
Calibrate Only Port Frequency
For Frequency conversion measurements, by default the reference receivers are calibrated for all resulting frequencies – even if they are currently not relevant for some ports. Using this switch, you can limit the calibrated frequencies to those frequencies that are required for the related port, which results in shorter calibration times.

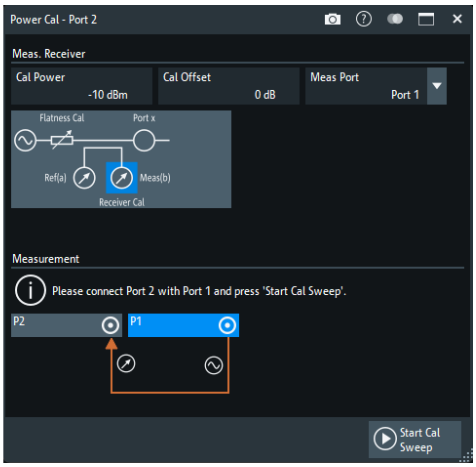
Power Cal dialog – Meas. Receiver

In "Meas. Receiver" mode, the "Power Cal" dialog guides you through the calibration of a measurement receiver. To ensure accurate source signal powers, a source power calibration must be performed before the measurement receiver calibration.

Background information
Refer to Section 4.5.6.3, "Measurement receiver calibration", on page 189.

Access: Power Cal wizard > "Meas. Receiver"





left: = source and meas. port identical
right = source and meas. port different

Port Overview
The dialog shows all receiver ports of the network analyzer. The selected port is displayed with the current cal power settings (see Section 5.11.3.2, "Cal Power Config dialog", on page 650); moreover, a circuit diagram visualizes the purpose of the measurement receiver calibration.

"Source Port" defines the type of measurement receiver calibration:

- If the source port is equal to the calibrated port, the measurement receiver is calibrated by the wave that is reflected back by a connected Open or Short standard. Connect the Open or Short standard to the calibrated port; no additional external test setup is required.

- If the source port and the calibrated port are different, the measurement receiver is calibrated by the wave generated at the source port. Connect the source port to the calibrated port, including any external devices that you used for the source power calibration.

For the source port, a source power calibration must be active.

Remote command:
`[SENSe<Ch>:]CORRection:POWer<PhyPt>:ACQuire`

Start Cal Sweep
Start the calibration sweep for the selected port and power calibration settings and close the dialog. The calibration is performed as described in "Calibration procedure" on page 190. No additional calibration settings are needed.

Remote command:
`[SENSe<Ch>:]CORRection:POWer<PhyPt>:ACQuire`
`[SENSe<Ch>:]CORRection:POWer:DATA`
`[SENSe<Ch>:]CORRection:POWer:DATA:PORT<PhyPt>`

Power Cal dialog – Source Flatness

In "Source Flatness" mode, the "Power Cal" dialog guides you through a source power calibration of a single port.

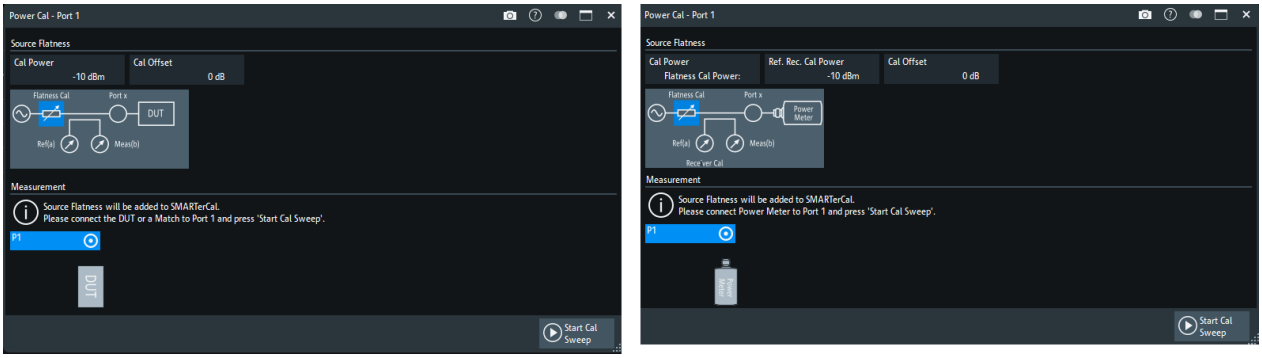
The dialog shows the selected source port with its current cal power settings (see Section 5.11.3.2, "Cal Power Config dialog", on page 650).

A circuit diagram visualizes the calibration setup, which depend on the selected Pwr Cal Method.

- "Ref. Receiver": connect a DUT or a Match to the related source port
- "Power Meter": connect a power meter to the related source port

Access: Power Cal wizard > "Source Flatness"

Background information
Refer to Section 4.5.6.2, "Source flatness calibration", on page 188 and Section 4.5.7.3, "Combining SMARTerCal with scalar power calibration", on page 195.



left = "Ref. Receiver" Pwr Cal Method
right = "Power Meter" Pwr Cal Method

Start Cal Sweep
Start the calibration sweeps for the selected port and power calibration settings and close the dialog. The calibration is performed as described in "Calibration procedure" on page 189.

Use the Power Cal Settings tab"Pwr Cal Settings" to modify the calibration procedure.

Remote command:
```
SOURce<Ch>:POWer:CORRection[:ACQuire]
SOURce<Ch>:POWer<PhyPt>:CORRection:COLLect[:ACQuire]
```

5.11.1.8 SMARTerCal wizard (Cal Unit)

The "SMARTerCal (Cal Unit)" wizard guides you through the setup and execution of an automatic "SMARTerCal".

Access: Channel – [Cal] > "Start Cal" > "SMARTerCal" – "Start (Cal Unit)"

The guided "SMARTerCal" is analogous to a regular automatic calibration with an additional power calibration sweep at a single analyzer port. It consists of the following steps:

1. **"Ports":** Select the calibrated ports, the cal unit characterization and the calibration type.
2. **"Connections":**
   a) Define the port assignment between the R&S ZNB and the calibration unit.
   b) Select one of the calibrated ports for power calibration.
3. **"SMARTerCal":**
   a) Acquire measurement data for all required standards for the selected system error calibration type.
   b) Apply the calculated error terms to the active channel.
   c) Acquire the power calibration data at the selected source port.
   d) Use the source power calibration data to correct the absolute receiver powers at all calibrated ports.

Background and related information

- Refer to Section 4.5.7, "SMARTerCal", on page 193 for background information
- Use the Power Cal wizard to perform an additional source flatness calibration (see Section 4.5.7.3, "Combining SMARTerCal with scalar power calibration", on page 195).

- When you apply the acquired "SMARTerCal", the active calibration is replaced and discarded.
  To persist any kind of calibration, you can transfer it to the calibration "Pool" using the Calibration Manager dialog.
- The system error correction data determined in a calibration procedure is stored in the analyzer. You can read these correction data using the remote control command `[SENSe<Ch>:]CORRection:CDATa`.
  If external switch matrices are involved, use `[SENSe<Ch>:]CORRection:SMATrix:CDATa` instead.

Step 1: Ports

Selects the calibrated analyzer ports, the cal unit characterization, the calibration type and the power meter connection.

Background information

Refer to the following sections:

- Section 4.5.7.2, "Calibration types", on page 194
- Section 4.5.5.3, "Characterization of calibration units", on page 179

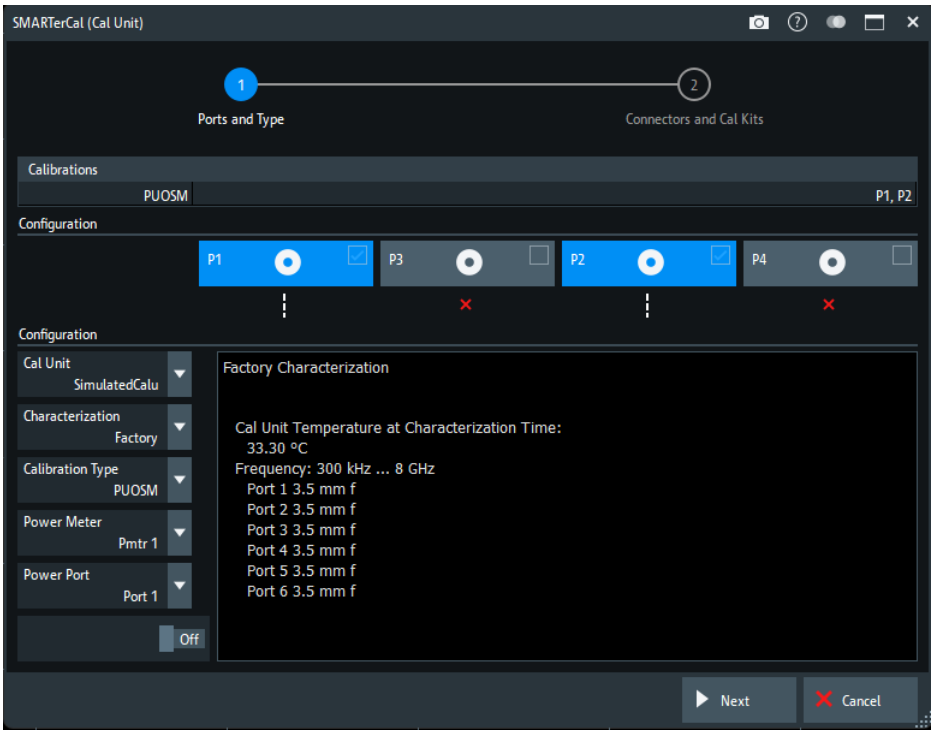


Figure 5-48: SMARTerCal (Cal Unit), Step 1: Ports

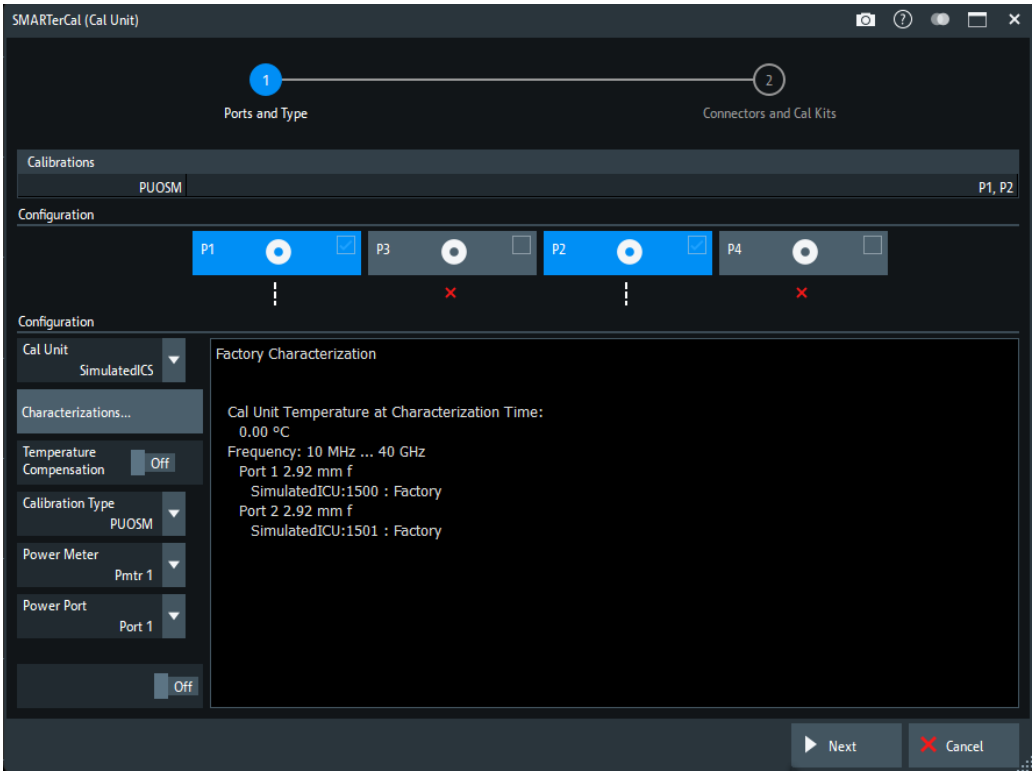


Figure 5-49: SMARTerCal (Cal Unit), Step 1: Ports (Inline Calibration System)

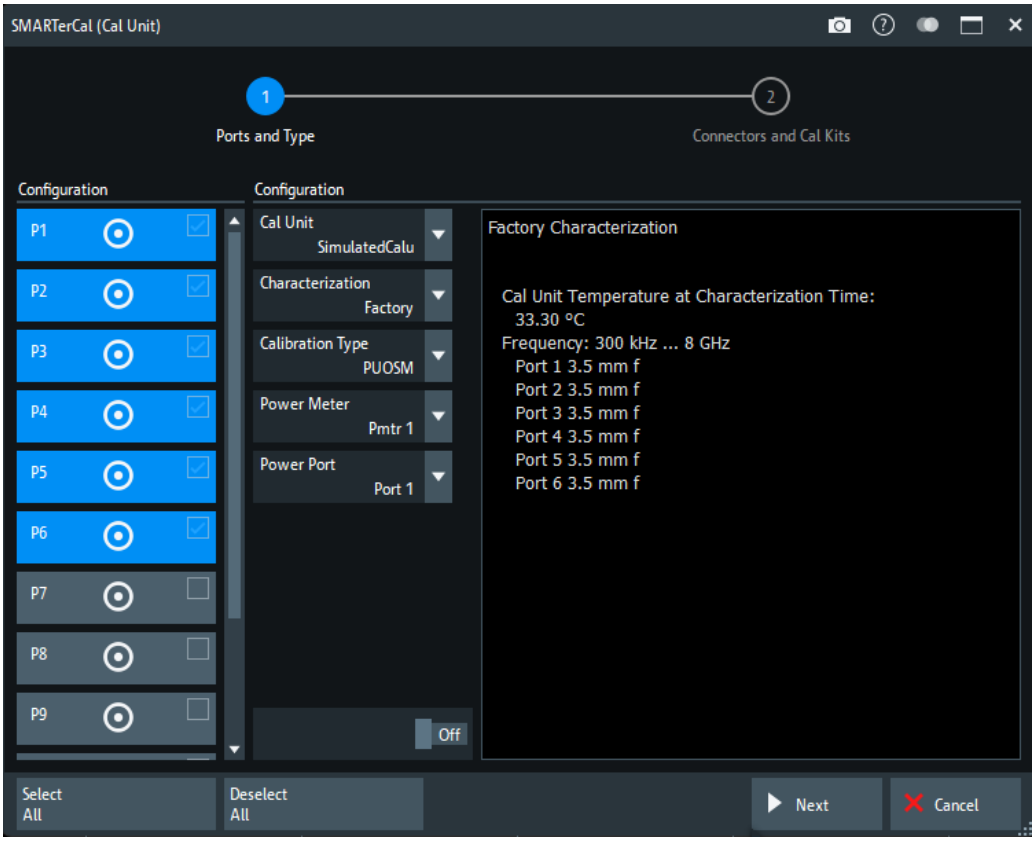


Figure 5-50: SMARTerCal (Cal Unit), Step 1: Ports (> 4 Ports)

Ports
Selects the test ports to be calibrated.

It is possible to select any combination of two or more test ports. If you are only interested in a single port *p*, perform a two-port SMARTerCal for a port pair including *p* (as Power Port).

Note: Calibration and port de-/activation.
The analyzer firmware automatically activates/deactivates ports during/after a (successful) calibration:

- Calibrated ports that were previously disabled, are automatically enabled as single-ended logical ports.
- An uncalibrated port that is not used by a measurement (i.e. the port is not required by any trace of the related channel) is disabled.

Remote command:
The port parameters in many calibration commands define ports to be calibrated.

Cal Unit
Displays the connected calibration units. The R&S ZNB auto-detects all calibration units which are connected to one of its USB ports. If several cal units are connected, one of them must be selected for calibration (active cal unit). A warning is displayed if the current sweep range of the network analyzer exceeds the characterized frequency range of the calibration unit.

For background information, see Section 4.5.5, "Automatic calibration", on page 175.

Remote command:
SYSTem:COMMunicate:RDEVice:AKAL:ADDRess:ALL?
SYSTem:COMMunicate:RDEVice:AKAL:ADDRess

Characterization
See "Characterization" on page 568.

Temperature Compensation
Available for inline calibration systems (ICS) only. Applies to all inline calibration units (ICUs) that are connected to the ICS and that are used with factory characterization data.

If enabled, the R&S ZNB firmware adjusts the factory characterization data according to the ICU temperatures.

Remote command:
[SENSe<Ch>:]CORRection:COLLect:AUTO:TEMPcomp

Calibration Type
Selects the calibration type for the selected physical ports. For an overview, refer to Section 4.5.7.2, "Calibration types", on page 194. The calibration types PUOSM and PTOSM are compatible with all port combinations.

Remote command:
[SENSe<Ch>:]CORRection:COLLect:AUTO:TYPE
[SENSe<Ch>:]CORRection:COLLect:AUTO:PORTs:TYPE
[SENSe<Ch>:]CORRection:COLLect:AUTO:CONFigure

Power Meter
Shows a list of all power meters that have been properly configured and are available for the scalar source power calibration of a single source port. The last configured power meter is selected by default. See "Configured Devices" on page 846.

Remote command:
[SENSe:]CORRection:COLLect:PMETer:ID

Power Port
Selects the port for the scalar source power calibration.

Source Flatness
If enabled, the "SMARTerCal" wizard opens the Power Cal wizard on completion.

There you can perform a complementary source flatness calibration at the ""SMARTer-Cal"ibrated" ports, and additional scalar power calibrations at other ports.

Select All/Deselect All
Selects/deselects all ports.

These buttons are only shown if more than 4 ports are available.

Next
Proceeds to Step 2: Connections. Next is unavailable (and a warning is displayed) if the following happens:

- Fewer than two ports have been selected.
- The selected characterization data do not cover all the ports to be calibrated.
- The selected characterization data do not contain all standards needed for the selected calibration type.

Step 2: Connections

Defines the port assignments between the R&S ZNB and the calibration unit.

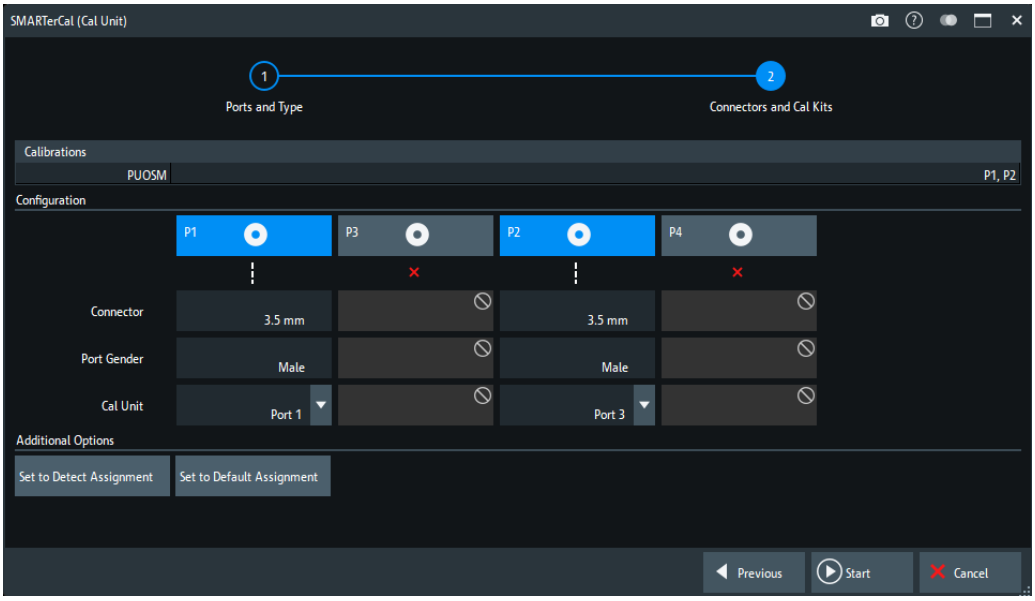


Figure 5-51: SMARTerCal (Cal Unit), Step 2: Connections (single)

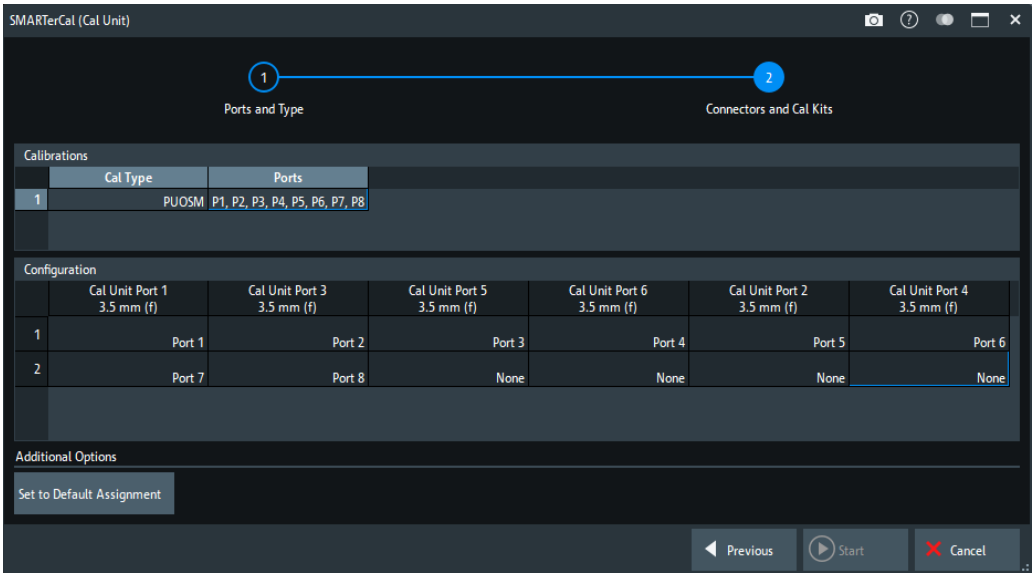


Figure 5-52: SMARTerCal (Cal Unit), Step 2: Connections

Related information

Refer to Section 4.5.5.6, "Multiple port assignments", on page 185 for details on multiple port assignments.

Optimized port assignment

If there is a switch matrix on the signal path, you can select between default and optimized port assignment (see "Optimized port assignment" on page 280)

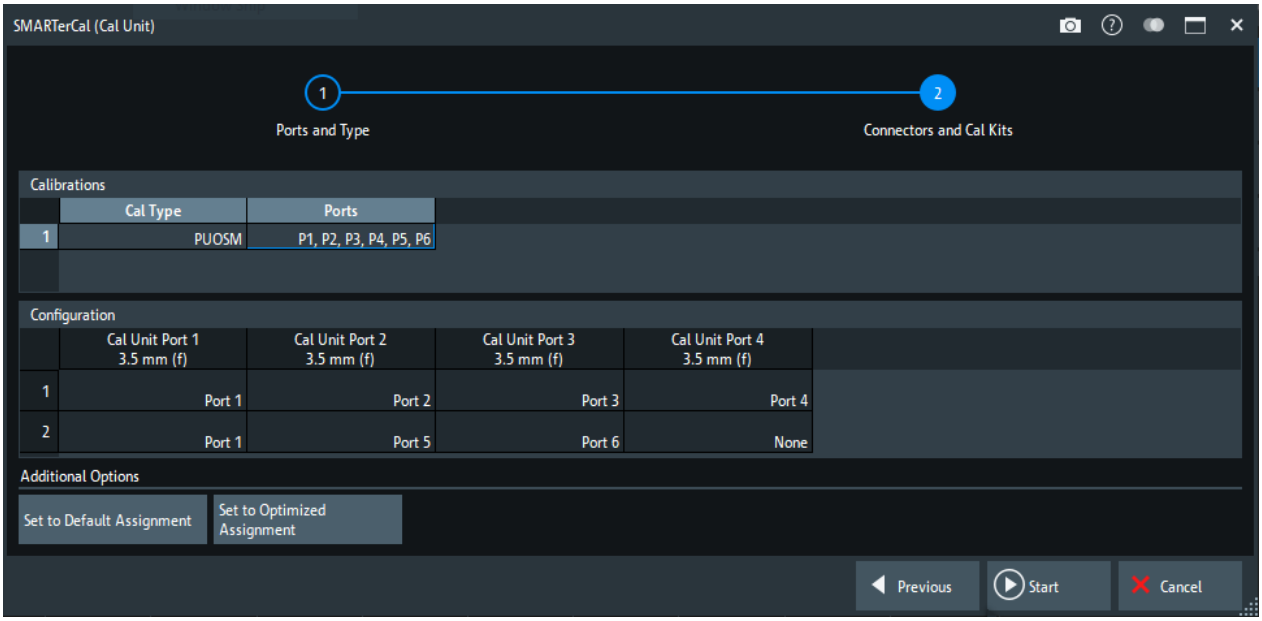


Figure 5-53: Optimized port assignment for 6-port R&S ZN-Z154

Port Assignment (manual)
The "Connections" step allows you to configure the assignments between VNA ports and calibration unit ports manually.

The R&S ZNB always proposes an optimum solution (minimum number of assignments) that also minimizes the physical port reconnections required between calibration stages. For user-modified assignments, it provides assistive information indicating insufficient, or redundant entries.

The test port connectors are automatically set according to the connector type of the selected calibration unit port.

Remote command:

```
[SENSe<Ch>:]CORRection:COLLect:AUTO:ASSignment<Asg>:DEFine
[SENSe<Ch>:]CORRection:COLLect:AUTO:ASSignment<Asg>:DEFine:TPORt
[SENSe<Ch>:]CORRection:COLLect:AUTO:ASSignment:COUNt?
```

Detect Assignment
Starts a procedure by which the R&S ZNB (with a little help from the attached calibration unit) auto-detects the connected ports. The automatic assignment replaces the configured one.

If auto-detection fails because of a high attenuation in the signal path, you can either enter the port assignment manually or connect matching port numbers and select "Default Port Assignment".

Remote command:
`[SENSe:]CORRection:COLLect:AUTO:PORTs:CONNection?`

Set to Default Port Assignment
Restores the default port assignments.

Remote command:
`[SENSe<Ch>:]CORRection:COLLect:AUTO:ASSignment:DEFine:DEFault`
`[SENSe<Ch>:]CORRection:COLLect:AUTO:ASSignment:DEFine:TPORt:DEFault`

Set to Optimized Assignment
Selects a port assignment that is optimized for measuring the passive load matches of the matrix test ports.

The optimized assignment is particularly recommended for calibration units with high through path losses, such as the R&S ZN-Z154. For other calibration units, usually it coincides with the default assignment, whose cabling can be a bit simpler.

If your measurement setup contains VNA ports that are not connected to a switch matrix, it is recommended to include (at least) one of them in the calibration. In the optimized port assignment, it is used as the source port for all passive load match measurements, and as the common port for all (unknown) Through measurements during calibration.

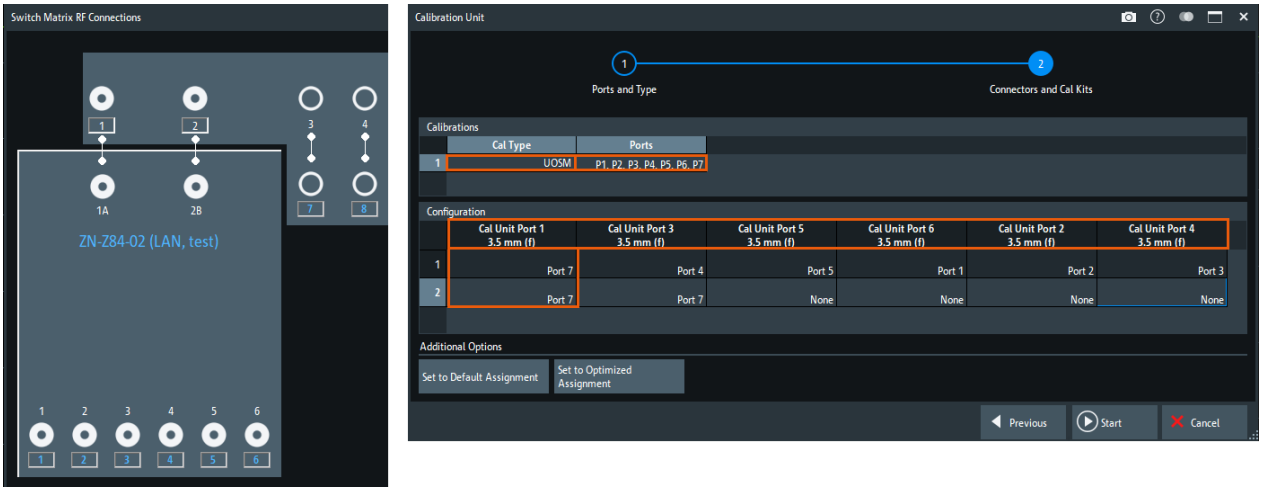


Figure 5-54: Direct VNA ports

Remote command:
`[SENSe<Ch>:]CORRection:COLLect:AUTO:ASSignment:DEFine:OPTimized`

Start
Proceeds to Step 3: SMARTerCal.

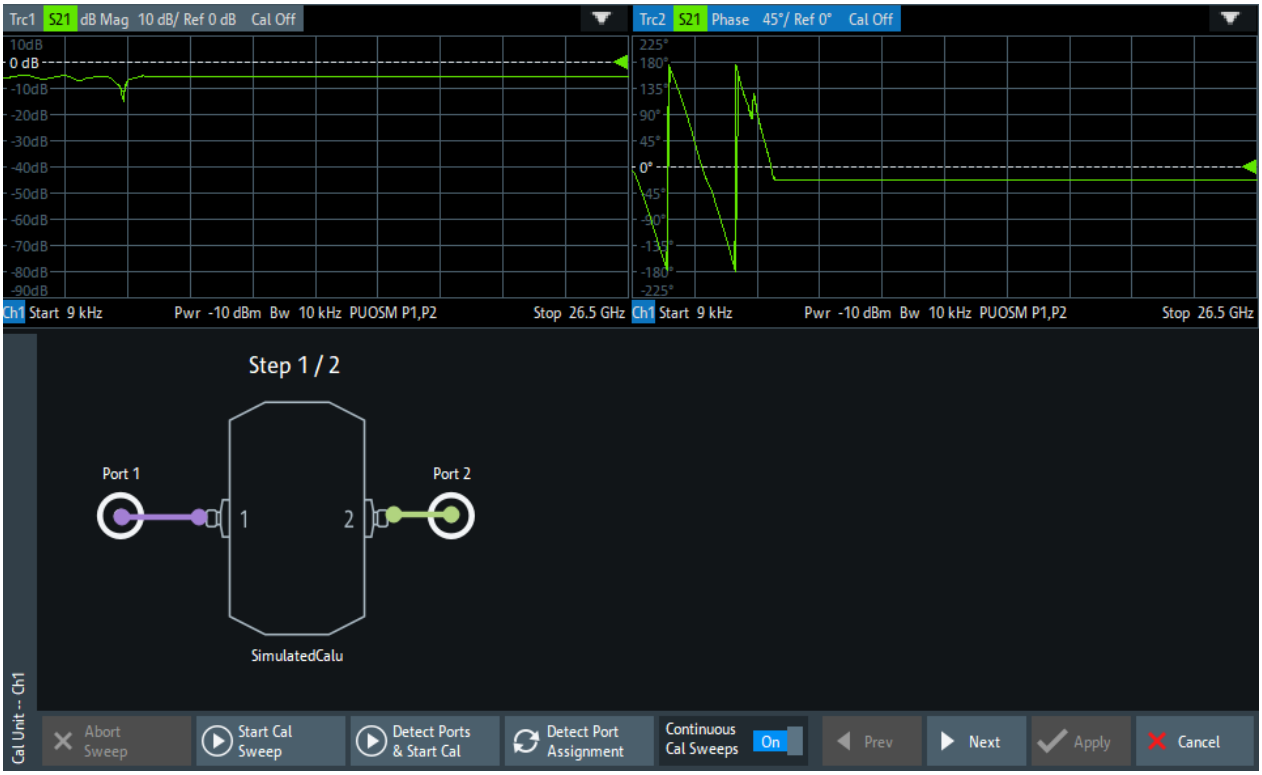
If the configured port assignments are invalid, this action is disabled.

Step 3: SMARTerCal

In the final step, the R&S ZNB displays a "SMARTerCal" screen that guides the user through the calibration data acquisition.

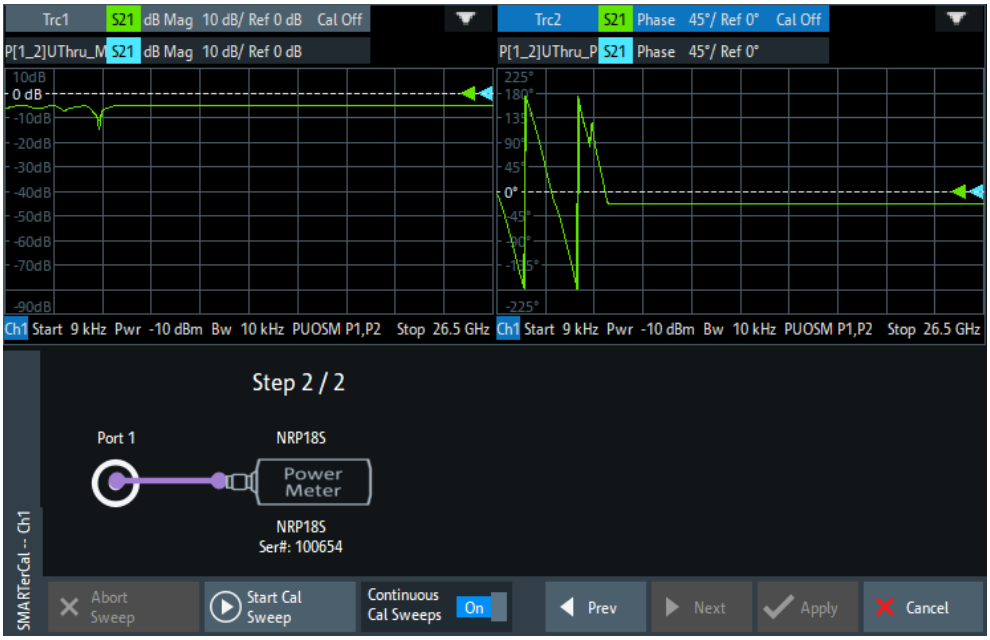
1. For each port assignment:
   a) Connect the calibration unit to the related set of test ports.
   b) Perform an automatic system error correction.

   The upper part of the "Cal Unit" screen displays the calibration sweep diagrams for the currently measured S-parameter. The lower part visualizes the active port assignment and the measurement progress.



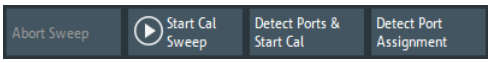
2. Finally:
   a) Replace the calibration unit by the power meter.
   b) Start the power calibration sweep.

   The upper part of the screen displays power trace diagrams and the lower part displays the power meter connection.



Use "Next"/"Prev" to navigate between the calibration substeps.

If you are calibrating a matrix setup and have selected "Set to Optimized Assignment" in the previous calibration step, the detection functionality slightly changes. Because a different cabling can invalidate the optimization logic, it is not possible to adjust the assignment to the auto-detected one. In this scenario, you can check the port assignment and continue with the calibration only if the check was successful.



Start Cal Sweep / Abort Sweep
Starts the necessary calibration sweeps or aborts them.

Note: The power calibration sweep is performed at the "Reference Receiver Cal Power" level specified in the Cal Power Config dialog.

A subsweep is performed for every required test port (pair), for every possible path (if external switch matrices are involved) and - during system error correction - for every required standard.

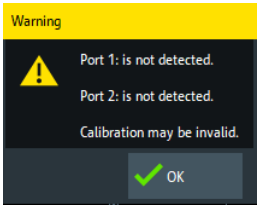
Remote command:

```
[SENSe<Ch>:]CORRection:COLLect:AUTO
[SENSe<Ch>:]CORRection:COLLect:AUTO:TYPE
[SENSe<Ch>:]CORRection:COLLect:AUTO:PORTs
[SENSe<Ch>:]CORRection:COLLect:AUTO:PORTs:TYPE
[SENSe<Ch>:]CORRection:COLLect:AUTO:ASSignment<Asg>:ACQuire
[SENSe<Ch>:]CORRection:COLLect:AUTO:POWer
```

Detect Ports & Start Cal
Performs the "Detect Assignment" and "Start Cal Sweep" functions, one after the other.

Detect Assignment
Starts a procedure by which the R&S ZNB (with a little help from the attached calibration unit) auto-detects the connected ports. The automatic assignment replaces the configured one.

In case auto-detection fails
- an error report is shown as a warning dialog
- the undetected port connections are marked with warning signs
- the calibration can be invalid



Remote command:
`[SENSe:]CORRection:COLLect:AUTO:PORTs:CONNection?`

Check Ports & Start Cal / Check Port Assignment
Only available for matrix setups, and if you have selected "Set to Optimized Assignment" in the previous calibration step.

"Check Port Assignment" checks whether the detected port connections coincide with the optimized assignment. "Check Ports & Start Cal" proceeds with the calibration if no deviation is found.

Remote command:
`[SENSe<Ch>:]CORRection:COLLect:AUTO:ASSignment<Asg>:CHECk?`
`[SENSe<Ch>:]CORRection:COLLect:AUTO:ASSignment<Asg>:ACQuire`

Continuous Cal Sweeps
See "Continuous Cal Sweeps" on page 814.

Prev/Next
Navigates between the port assignments. Only available if multiple port assignments are required.

Apply
Is enabled when sufficient data have been acquired for the calibrated ports and standards and for the power meter. The button starts the calculation of the calibration data and closes the calibration wizard.

The results are applied to the active channel (or to all channels in the active recall set, if all channels are calibrated).The current instrument settings are stored with the correction data.

To avoid incompatibilities, older calibration data is deleted unless it has been transferred into the calibration "Pool" using the Calibration Manager dialog.

Use the Power Cal wizard to perform an additional source flatness calibration (see Section 4.5.7.3, "Combining SMARTerCal with scalar power calibration", on page 195).

Note: Checks during the calculation of correction data

Incompatibilities between the selected calibration type, the standards and the channel settings can cause the calibration to be inaccurate. The analyzer auto-detects potential sources of errors and displays appropriate, self-explanatory notice boxes.

Remote command:

```
[SENSe<Ch>:]CORRection:COLLect:AUTO
[SENSe<Ch>:]CORRection:COLLect:AUTO:TYPE
[SENSe<Ch>:]CORRection:COLLect:AUTO:PORTs
[SENSe<Ch>:]CORRection:COLLect:AUTO:PORTs:TYPE
[SENSe<Ch>:]CORRection:COLLect:AUTO:SAVE
[SENSe<Ch>:]CORRection:COLLect:AUTO:POWer
```

5.11.1.9 SMARTerCal wizard (Cal Unit, MultiCal)

The "MultiCal" version of the SMARTerCal wizard (Cal Unit) guides you through the setup and simultaneous acquisition of multiple SMARTerCals.

Access: Channel – [Cal] > "Start Cal" > "SMARTerCal" – "Start (Cal Unit)"

The guided SMARTerCal is analogous to a regular automatic calibration with an additional power calibration sweep at a single analyzer port. It consists of the following steps:

1. **"Ports":** Define the SMARTerCals to be performed (test ports, calibration unit, characterization, calibration type, power meter, power port)
2. **"Connections":** Define the port assignments between the R&S ZNB and the calibration units.
3. **"SMARTerCal":**
   a) For each configured SMARTerCal and port assignment, acquire error correction data for the required calibration standards (provided by the selected calibration unit).
   b) Apply the resulting error terms to the active channel.
   c) Acquire the power calibration data at the selected source ports.
   d) Use this result to correct the absolute receiver powers at all calibrated ports.



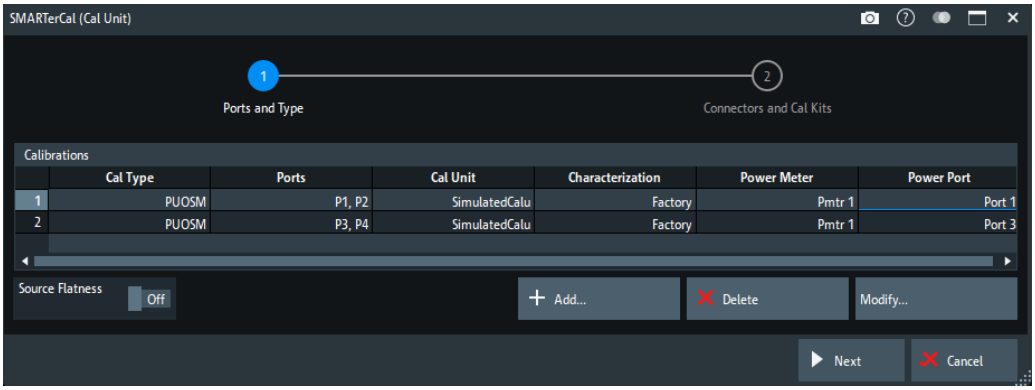
Background and related information

- The "Multiple Cal in Calibration Wizard" feature is disabled by default and has to be activated in the Calibration tab of the "System Config" dialog.
- Use the Power Cal wizard to perform an additional source flatness calibration (see Section 4.5.7.3, "Combining SMARTerCal with scalar power calibration", on page 195).
- Refer to Section 5.11.1.8, "SMARTerCal wizard (Cal Unit)", on page 601 for further information.

Step 1: Ports

The first page of the wizard presents the list of already configured calibrations and allows you to:

- "Add" new calibrations
- "Delete" or "Modify" existing calibrations



Source Flatness
If enabled, the "SMARTerCal" wizard opens the Power Cal wizard on completion.

There you can perform a complementary source flatness calibration at the ""SMARTer-Cal"ibrated" ports, and additional scalar power calibrations at other ports.

Add
Adds a new system error correction.

The "Add" button opens the Define Calibration dialog without pre-selected ports.

Delete
Deletes the selected calibration from the list of configured calibrations.

Modify
Edits the selected system error correction: opens the Define Calibration dialog with the corresponding ports and calibration type pre-selected.

Next
Proceeds to Step 2: Connections.

Inactive as long as no SMARTerCal is defined.

Define Calibration Dialog

In MultiCal mode, the "Define Calibration" dialog allows you to define the individual SMARTerCals: select the test ports to be calibrated, the calibration type, the calibration unit and characterization, the power meter and power port.

Access: SMARTerCal wizard (Cal Unit, MultiCal) > Step 1: "Ports" > "Add" / "Modify"

- The "Multiple Cal in Calibration Wizard" (MultiCal) feature is disabled by default and has to be activated in the Calibration tab of the "System Config" dialog.
- For background information on calibration types, see Section 4.5.1, "Calibration types", on page 155.

The layout of the main panel depends on the number of test ports available.

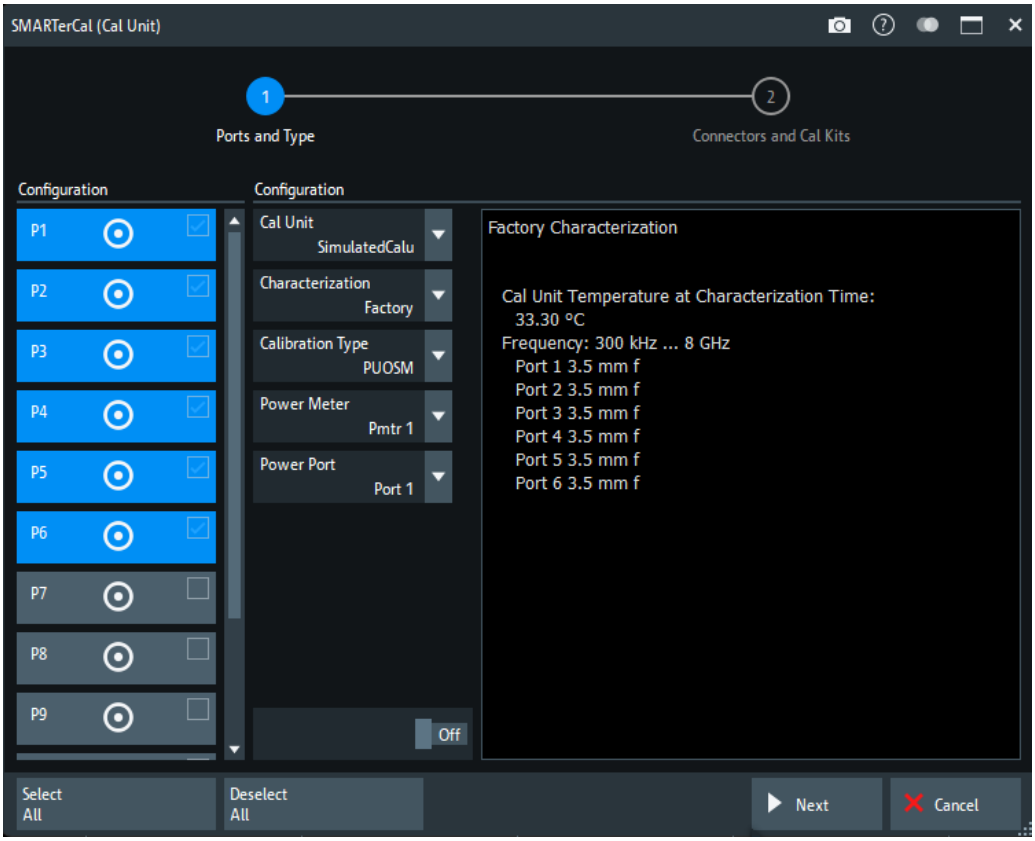


Figure 5-55: Define Calibration dialog (MultiCal, > 4 test ports)

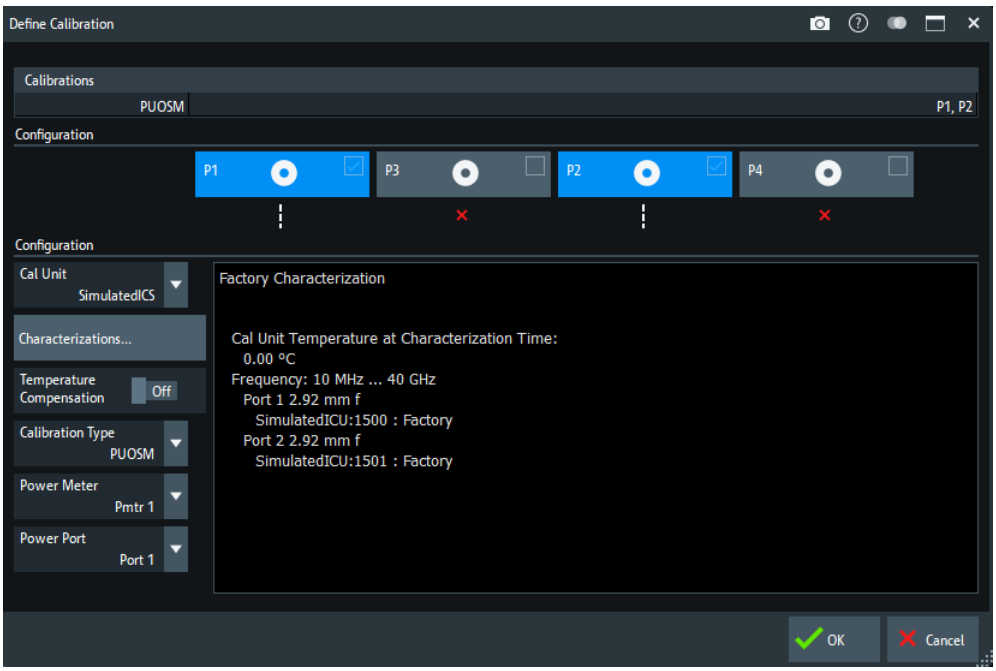


Figure 5-56: Define Calibration dialog (MultiCal, Inline Calibration System)

Ports
Selects the test ports to be calibrated.

It is possible to select any combination of two or more test ports. If you are only interested in a single port *p*, perform a two-port SMARTerCal for a port pair including *p* (as Power Port).

Note: Calibration and port de-/activation.
The analyzer firmware automatically activates/deactivates ports during/after a (successful) calibration:

- Calibrated ports that were previously disabled, are automatically enabled as single-ended logical ports.
- An uncalibrated port that is not used by a measurement (i.e. the port is not required by any trace of the related channel) is disabled.

Remote command:
The port parameters in many calibration commands define ports to be calibrated.

Cal Unit
Displays the connected calibration units. The R&S ZNB auto-detects all calibration units which are connected to one of its USB ports. If several cal units are connected, one of them must be selected for calibration (active cal unit). A warning is displayed if the current sweep range of the network analyzer exceeds the characterized frequency range of the calibration unit.

For background information, see Section 4.5.5, "Automatic calibration", on page 175.

Remote command:
`SYSTem:COMMunicate:RDEVice:AKAL:ADDRess:ALL?`
`SYSTem:COMMunicate:RDEVice:AKAL:ADDRess`

Characterization
See "Characterization" on page 568.

Temperature Compensation
Available for inline calibration systems (ICS) only. Applies to all inline calibration units (ICUs) that are connected to the ICS and that are used with factory characterization data.

If enabled, the R&S ZNB firmware adjusts the factory characterization data according to the ICU temperatures.

Remote command:
`[SENSe<Ch>:]CORRection:COLLect:AUTO:TEMPcomp`

Calibration Type
Selects the calibration type for the selected physical ports. For an overview, refer to Section 4.5.7.2, "Calibration types", on page 194. The calibration types PUOSM and PTOSM are compatible with all port combinations.

Remote command:
`[SENSe<Ch>:]CORRection:COLLect:AUTO:TYPE`
`[SENSe<Ch>:]CORRection:COLLect:AUTO:PORTs:TYPE`
`[SENSe<Ch>:]CORRection:COLLect:AUTO:CONFigure`

Power Meter
Shows a list of all power meters that have been properly configured and are available for the scalar source power calibration of a single source port. The last configured power meter is selected by default. See "Configured Devices" on page 846.

Remote command:
`[SENSe:]CORRection:COLLect:PMETer:ID`

Power Port
Selects the port for the scalar source power calibration.

Select All/Deselect All
Selects/deselects all ports.

These buttons are only shown if more than 4 ports are available.

Step 2: Connections

Defines the port assignments between the R&S ZNB and the calibration unit.

Related information
Refer to Section 4.5.5.6, "Multiple port assignments", on page 185 for details on multiple port assignments.

The layout of the main panel depends on the number of test ports available.

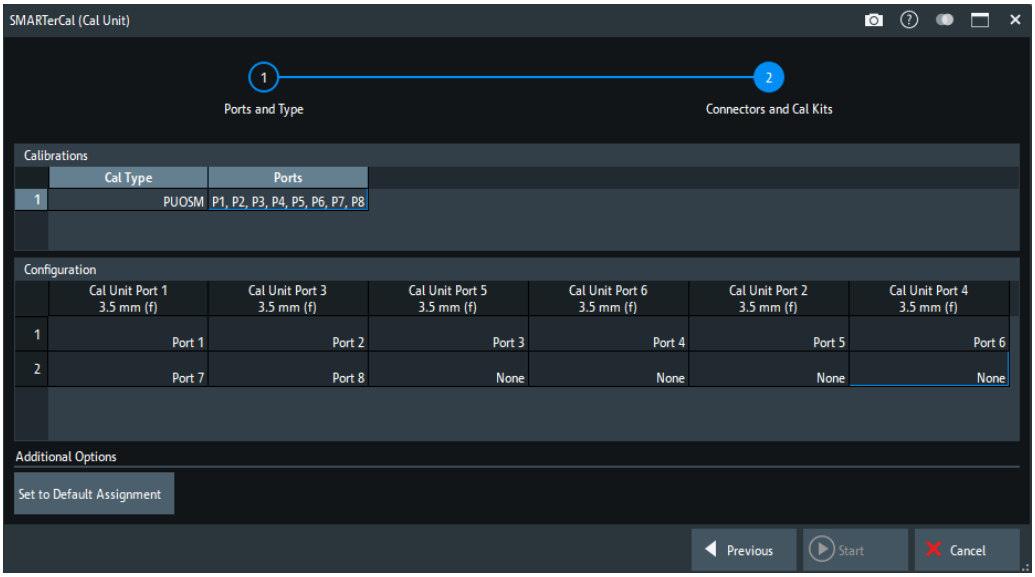


Figure 5-57: Multiple port assignments

Similar functionality as for the "SingleCal" version of the wizard (see "Step 2: Connections" on page 606): select the adequate calibration in the "Cal Type"/"Ports" table to display and edit the corresponding port assignments.

Start
Proceeds to Step 3: SMARTerCal.

If the configured port assignments are invalid, this action is disabled.

Step 3: SMARTerCal

In the final step, the R&S ZNB displays a "SMARTerCal" screen that guides the user through the calibration data acquisition.

The logic is identical to that of the single calibration case (see "Step 3: SMARTerCal" on page 609).

5.11.1.10 SMARTerCal wizard (manual)

The "SmarterCal (Manual)" wizard guides you through the setup and execution of a manual SMARTerCal.

Access: Channel – [Cal] > "Start Cal" > "SMARTerCal" – "Start (Manual)"

The wizard proceeds through the following steps:

1. **"Ports and Type":** Select the ports to be calibrated, the calibration type, a connected power meter and the "Power Port".
2. **"Connectors and Cal Kits":** Specify the connector type and gender of the selected ports. If necessary, load or change a calibration kit.
3. **"SMARTerCal":**

a) Acquire error correction data for the required ports and port pairs, and the required standards.
b) Acquire source power calibration data at the "Power Port".
c) Finally, decide whether to apply the resulting calibration.

Background and related information

- Refer to Section 4.5.7, "SMARTerCal", on page 193 for background information.
- See Section 5.11.1.11, "SMARTerCal wizard (manual, MultiCal)", on page 627 for the "MultiCal" variant of the wizard.
- Use the Power Cal wizard to perform an additional source flatness calibration (see Section 4.5.7.3, "Combining SMARTerCal with scalar power calibration", on page 195).

- If the active channel is already calibrated, the wizard attempts to load the calibration. If the calibration setup is unchanged and sweep data are available, the existing system error correction can be optimized without repeating the measurement of all standards.
- When you apply the acquired SMARTerCal, the active calibration is replaced and discarded.
  To persist any kind of calibration, you can transfer it to the cal "Pool" using the Calibration Manager dialog.
- The active system error correction data can be read (and modified) using the remote control command `[SENSe<Ch>:]CORRection:CDATa`.
  If external switch matrices are involved, use `[SENSe<Ch>:]CORRection:SMATrix:CDATa` instead.

Step 1: Ports and Type

Selects the calibrated analyzer ports, the calibration type and specifies the power meter connection.

Background information

Refer to Section 4.5.7.2, "Calibration types", on page 194.

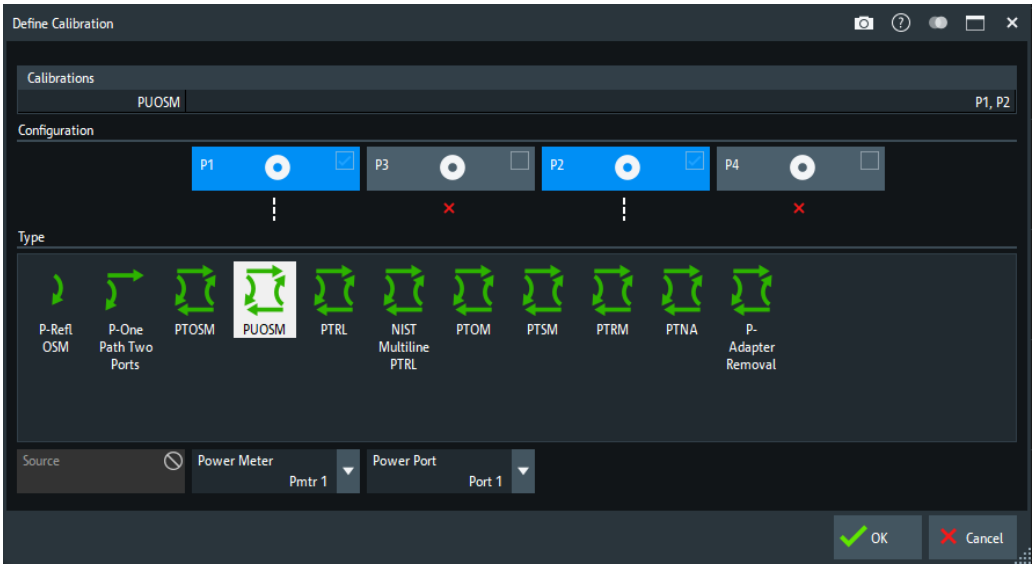


Figure 5-58: SMARTerCal (Manual), Step 1: Ports and Type

The layout of the main panel depends on the number of test ports available.



Figure 5-59: SMARTerCal(Manual), Step 1: Ports and Type (>4 ports)

Ports
Selects the test ports to be calibrated.

It is possible to select any combination of two or more test ports. If you are only interested in a single port *p*, perform a two-port SMARTerCal for a port pair including *p* (as Power Port).

Note: Calibration and port de-/activation.

The analyzer firmware automatically activates/deactivates ports during/after a (successful) calibration:

- Calibrated ports that were previously disabled, are automatically enabled as single-ended logical ports.
- An uncalibrated port that is not used by a measurement (i.e. the port is not required by any trace of the related channel) is disabled.

Remote command:
The port parameters in many calibration commands define ports to be calibrated.

Type
Selects the calibration type to be performed. For background information, see Section 4.5.7.2, "Calibration types", on page 194.

SMARTerCal is based on a "One Path Two Ports" or a full n-port calibration (n > 1): "Adapter Removal" is a 2-port calibration, all other calibration types can be used for any combination of 2 or more ports.

Remote command:
`[SENSe<Ch>:]CORRection:COLLect:METHod:DEFine`

Power Meter
Shows a list of all power meters that have been properly configured and are available for the scalar source power calibration of a single source port. The last configured power meter is selected by default. See "Configured Devices" on page 846.

Remote command:
`[SENSe:]CORRection:COLLect:PMETer:ID`

Power Port
Selects the port for the scalar source power calibration.

Source Flatness
If enabled, the "SMARTerCal" wizard opens the Power Cal wizard on completion.

There you can perform a complementary source flatness calibration at the ""SMARTerCal"ibrated" ports, and additional scalar power calibrations at other ports.

Select All/Deselect All
Selects/deselects all ports.

These buttons are only shown if more than 4 ports are available.

Next
Proceeds to "Step 2: Connectors and Cal Kits" on page 620. This button is inactive (and a warning is displayed) if fewer than two ports are selected.

Step 2: Connectors and Cal Kits

Selects the connector type and gender for all ports and allows you to import a calibration kit.

Background information

Refer to Section 4.5.2, "Calibration standards and calibration kits", on page 168.

Messages in the dialog

An information message (or error message) is displayed, if one of the following happens:

- One of the selected calibration kits is described by ideal kit parameters or typical values.
- One of the selected calibration kits does not contain all standards that are required for the previously selected calibration type.
- Different connector types are defined at the ports but the selected calibration type requires uniform connectors.
- A cal kit standard does not cover the entire calibrated frequency range.
- The selected power meter does not cover the entire calibrated frequency range.

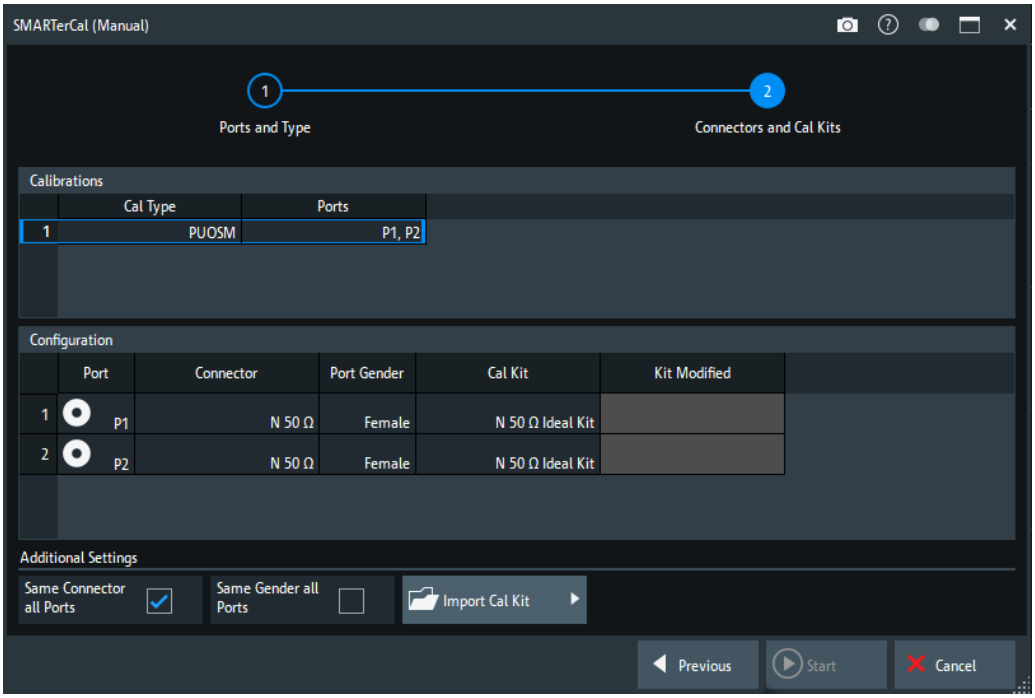


Figure 5-60: SMARTerCal(Manual), Step 2: Connectors and Cal Kits

The layout of the main panel depends on the number of test ports available.

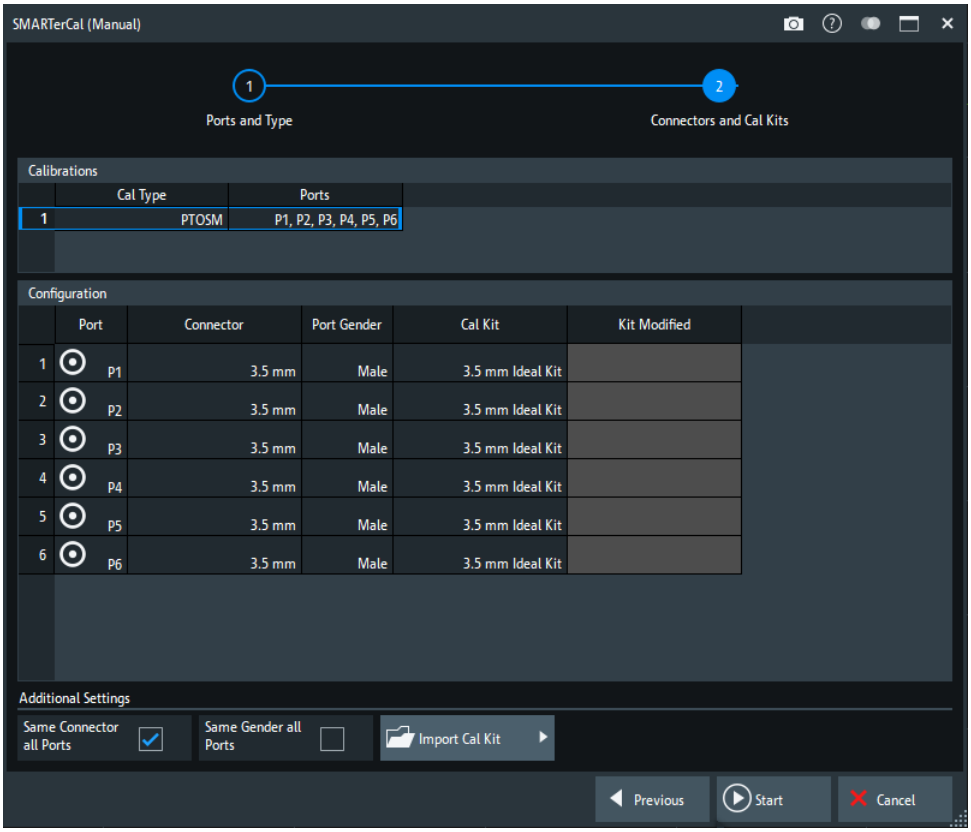


Figure 5-61: SMARTerCal (Manual), Step 2: Connectors and Cal Kits (>4 ports)

The upper part of the panel shows the selected ports and the calibration type. The lower part gives access to the connector and cal kit settings.

Connector / Port Gender
Defines the connector types and genders of the ports to be calibrated. For symmetric (sexless) connector types (e.g. 7 mm / PC7), "Port Gender" is unavailable.

If "Same Connector All Ports" is active, the connector types at all ports (but not their gender) are always adjusted to the current selection. If "Same Gender All Ports" is active, the genders at all ports are always adjusted to the current selection.

User-defined connectors can be added or removed in the Cal Connector Types dialog.

Remote command:
`[SENSe<Ch>:]CORRection:COLLect:CONNection<PhyPt>`
`[SENSe<Ch>:]CORRection:COLLect:SCONnection<PhyPt>`
`[SENSe:]CORRection:CONNection`
`[SENSe:]CORRection:CONNection:CATalog?`
`[SENSe<Ch>:]CORRection:CONNection:DELete`

Cal Kit
Selects a cal kit for the connector at each selected physical port. The drop-down list contains all available calibration kits for the selected connector type.

The assignment of a calibration kit to a connector type must be the same for all physical ports: If a calibration kit is changed, the R&S ZNB automatically assigns the new kit to all ports with the same connector type.

Use "Import Cal Kit" to add new kits to the list.

Remote command:
`[SENSe:]CORRection:CKIT:SELect`

Same Connector All Ports / Same Gender All Ports
Assigns the same connector type or gender to all selected physical ports. For some multi-port calibration types, the port connector types must be equal, e.g. because they require a Through standard with known characteristics.

Remote command:
`[SENSe<Ch>:]CORRection:COLLect:CONNection:PORTs`
`[SENSe<Ch>:]CORRection:COLLect:CONNection:GENDers`

Import Cal Kit
Opens the "Import Calibration Kit" dialog that allows you to import a cal kit file. For background information, see Section 4.5.2.4, "Cal kit Files", on page 173.

Remote command:
`MMEMory:LOAD:CKIT`

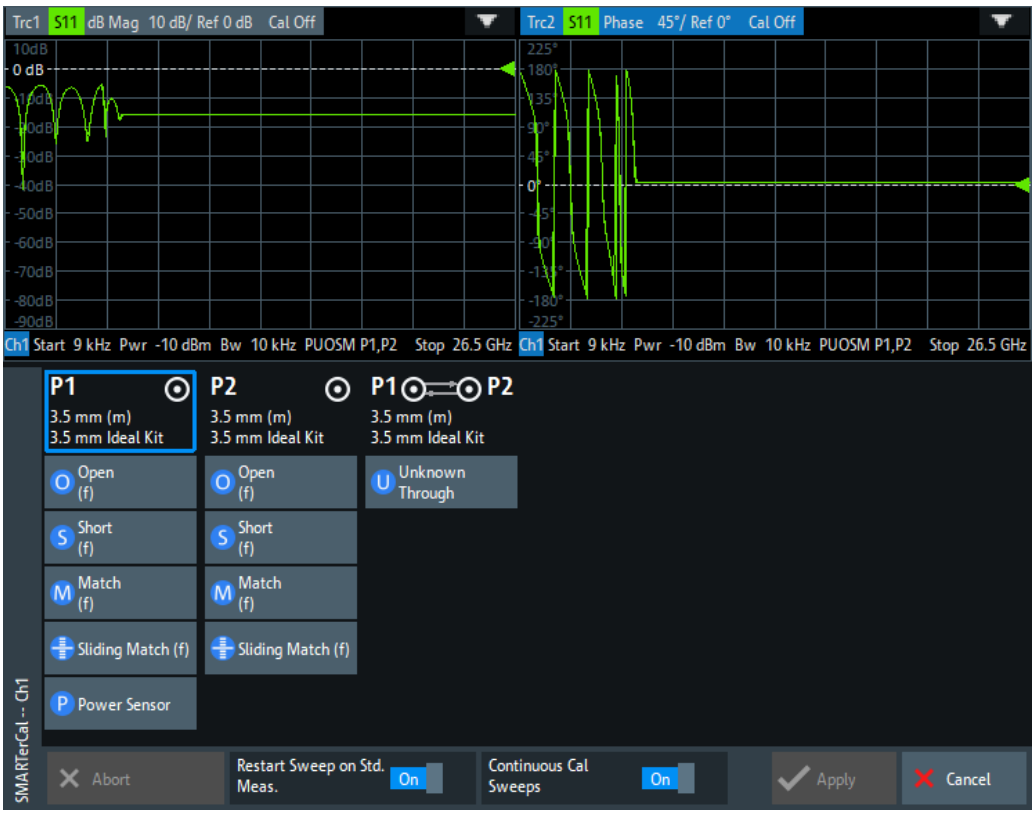
Back
Go back to Step 2: Connectors and Cal Kits.

Start
Start Step 3: SMARTerCal.

Step 3: SMARTerCal

Allows you to acquire error correction data for every required port (pair) and calibration standard, plus power correction data from the power meter. The order of the system error correction and power calibration sweeps is arbitrary; ensure that you always connect the proper equipment.

On "Apply", the R&S ZNB calculates the system error correction (error terms) from the measurement data of the standards and uses the power meter result to correct the absolute receiver powers. Finally it applies the SMARTerCal results to the active channel.



The upper part of the calibration screen presents

- calibration sweep diagrams for the currently measured S-parameter during standards measurement,
- power trace diagrams during the power calibration sweep

The lower part displays the calibrated port (pairs) with their related measurements and visualizes the measurement progress.

Calibration Sweep Diagrams

During the calibration sweep, each diagram contains a single S-parameter trace and a typical result trace for the measured calibration standard.

If switch matrices are involved, a sweep is performed for every possible signal path and for each of these paths a separate S-parameter trace is shown (see Section 4.7.30.5, "Multiport calibration", on page 277).

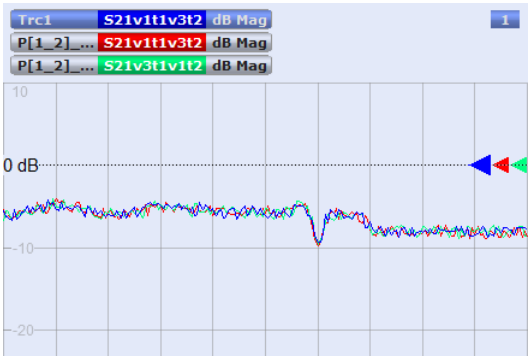


Figure 5-62: Multiple traces due to multiple paths

The purpose of the typical result traces "Trc1"" and "Trc2" is to avoid connection errors and to track hardware problems: if the correct standard type is measured, and everything is properly connected, then the measured traces are expected to be similar to the typical trace.

The S-Parameter traces are labeled P[*j*_*i*]_<standard type> S*ij*, where *j* indicates the input (test) port and *i* indicates the output port, e.g. P[1_2]_Unknown_Through S21.

If switch matrices are involved, the label also indicates:
- which VNA port *b* is switched to the input port *j* and
- which VNA port *a* is switched to the output port *i*

Instead of a single S*ij* trace we have multiple traces S*ij*v*a*t*i*v*b*t*j*. E.g., "S21v1t1v2t2" means that S21 is measured with VNA port 1 connected to the input port and VNA port 2 connected to the output port.

Power Sweep Diagrams
The "Pmtr<n>(P<m>)" traces are displayed during the "Power" calibration sweep.

Ports, Standards and Power
Allows you to perform the required calibration standard measurements, plus the additional "Power" measurement.

A green checkmark indicates that the calibration data of a standard or power meter has been acquired successfully. A green checkmark after the port symbol indicates that the minimum number of calibration measurements for the port has been performed.

Tip: Optional calibration measurements

- If the selected calibration kit comprises a Sliding Match, then for every required Match measurement either the Match or at least three positions of the Sliding Match must be measured. See Section 4.5.2.3, "Sliding Match standards", on page 172.
- For a PTRL calibration, at least one line standard must be measured between any pair of calibrated ports. See Section 4.5.1.9, "TRL calibration", on page 163.

Note: The power calibration sweep is performed at the "Reference Receiver Cal Power" setting from the scalar power calibration; see "Reference Receiver Cal Power" on page 652.

Remote command:
[SENSe<Ch>:]CORRection:COLLect[:ACQuire]:SELected
SOURce<Ch>:POWer:CORRection[:ACQuire]
See also: [SENSe<Ch>:]CORRection:COLLect:LOAD:SELected

Restart Sweep on Std. Meas.
If this function is active, a new standard measurement initiates a new sweep, starting at the beginning ("Start") of the sweep range: The sweep points for the calibration sweep are in ascending order, like for an ordinary measurement.

If "Restart Sweep on Std. Meas." is inactive, the new standard measurement is started at the current sweep point; the current sweep is continued as a calibration sweep.

Note:
This function has a secondary effect in IDLE mode (i.e. while the calibration sweep is NOT running):
- If active, only the selected trace is refreshed.
- If inactive, the R&S ZNB permanently refreshes all traces of all diagrams which can put a heavy load on the connected switch matrices (if any).

Hence it is recommended (and default) to activate it, in particular if one of the matrices uses *mechanical* switches (that wear off apart from making noise).

Continuous Cal Sweeps
See "Continuous Cal Sweeps" on page 814.

Apply
Is enabled when sufficient data have been acquired for the calibrated ports and standards and for the power meter. The button starts the calculation of the calibration data and closes the calibration wizard. The current instrument settings are stored with the correction data.

To avoid incompatibilities, older calibration data is deleted unless it has been transferred into the "Cal Pool" using the "Calibration Manager".

Use the Power Cal wizard to perform an additional source flatness calibration (see Section 4.5.7.3, "Combining SMARTerCal with scalar power calibration", on page 195).

Note: Checks during the calculation of correction data
Incompatibilities between the selected calibration type, the standards and the channel settings can cause the calibration to be inaccurate. The analyzer auto-detects potential sources of errors and displays appropriate, self-explanatory notice boxes.

Remote command:
[SENSe<Ch>:]CORRection:COLLect:SAVE:SELected[:DUMMy]
[SENSe<Ch>:]CORRection:COLLect:SAVE:SELected:DEFault
[SENSe<Ch>:]CORRection:COLLect:DELete
[SENSe<Ch>:]CORRection:DATA:PARameter<Sfk>?
[SENSe<Ch>:]CORRection:DATA:PARameter<Sfk>:PORT<PhyPt>?
[SENSe<Ch>:]CORRection:DATE?
[SENSe<Chn>:]CORRection:SSTate?
[SENSe<Ch>:]CORRection:STIMulus?
[SENSe<Ch>:]CORRection:STIMulus:PORT<PhyPt>?

5.11.1.11 SMARTerCal wizard (manual, MultiCal)

The multiple calibration version of the SMARTerCal wizard (manual) guides you through the setup and simultaneous manual acquisition of multiple SMARTerCals. Except for the possibility to create multiple calibrations, the layout and functionality is the same as for the single calibration version of the wizard.

Access: Channel – [Cal] > "Start Cal" > "SMARTerCal" – "Start... (Manual)"

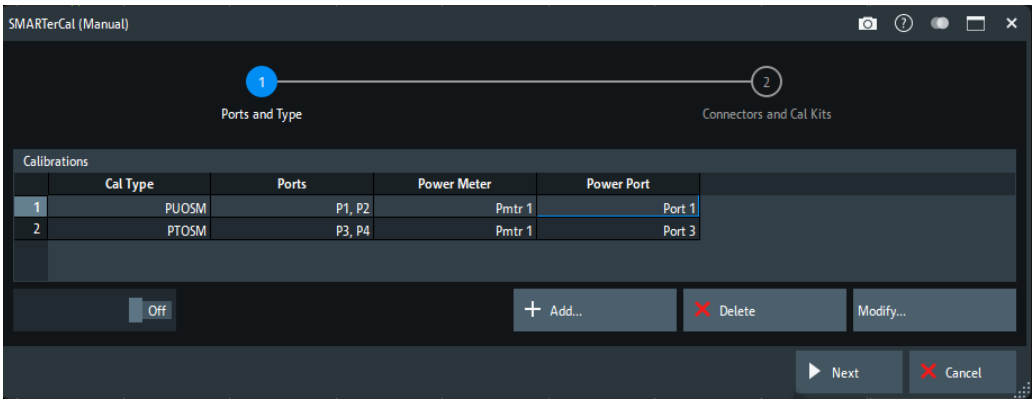
Background and related information

- The "Multiple Cal in Calibration Wizard" feature is disabled by default and has to be activated in the Calibration tab of the "System Config" dialog.
- See Section 5.11.1.10, "SMARTerCal wizard (manual)", on page 617 for more information.

Step 1: Ports and Type

The first page of the wizard presents the list of already configured SMARTerCals and allows you to:

- "Add" new SMARTerCals
- "Delete" or "Modify" existing SMARTerCals



Source Flatness
If enabled, the "SMARTerCal" wizard opens the Power Cal wizard on completion.

There you can perform a complementary source flatness calibration at the ""SMARTer-Cal"ibrated" ports, and additional scalar power calibrations at other ports.

Add
Adds a new SMARTerCal.

The "Add" button opens the Define Calibration dialog without pre-selected ports.

Delete
Deletes the selected SMARTerCal from the list of configured SMARTerCals.

Modify
Edits the selected SMARTerCal: opens the Define Calibration dialog with the corresponding ports and calibration type pre-selected.

Next
Proceeds to Step 2: Connectors and Cal Kits.

Inactive as long as no calibration is defined.

Define Calibration dialog

Selects the calibrated analyzer ports, the calibration type and specifies the power meter connection.

Background information
Refer to Section 4.5.7.2, "Calibration types", on page 194.

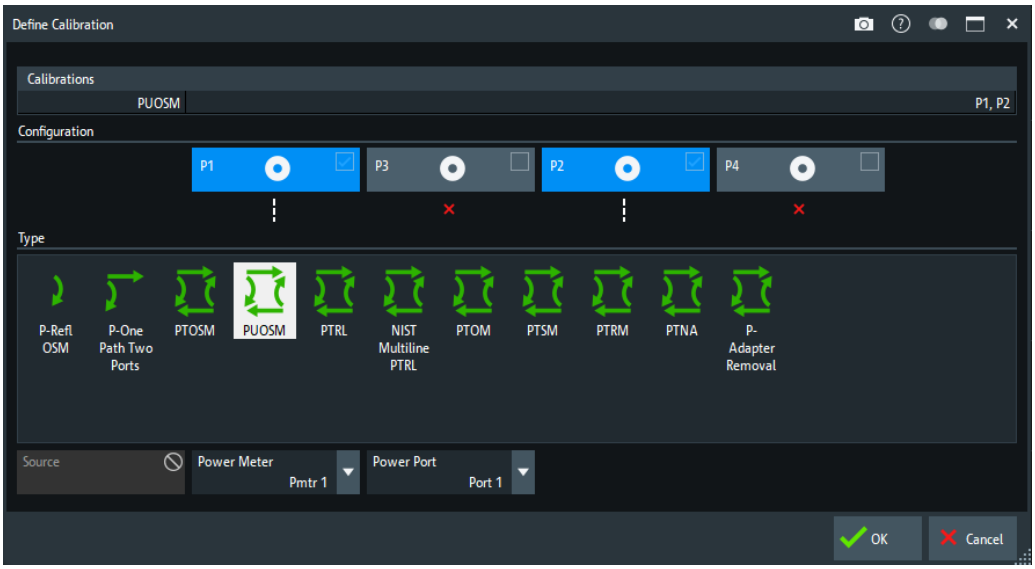


Figure 5-63: Define Calibration dialog (SMARTerCal)

The layout of the main panel depends on the number of test ports available.



Figure 5-64: Define Calibration dialog (SMARTerCal, >4 ports)

Ports
Selects the test ports to be calibrated.

It is possible to select any combination of two or more test ports. If you are only interested in a single port *p*, perform a two-port SMARTerCal for a port pair including *p* (as Power Port).

Note: Calibration and port de-/activation.
The analyzer firmware automatically activates/deactivates ports during/after a (successful) calibration:

- Calibrated ports that were previously disabled, are automatically enabled as single-ended logical ports.
- An uncalibrated port that is not used by a measurement (i.e. the port is not required by any trace of the related channel) is disabled.

Remote command:
The port parameters in many calibration commands define ports to be calibrated.

Type
Selects the calibration type to be performed. For background information, see Section 4.5.7.2, "Calibration types", on page 194.

SMARTerCal is based on a "One Path Two Ports" or a full n-port calibration (n > 1): "Adapter Removal" is a 2-port calibration, all other calibration types can be used for any combination of 2 or more ports.

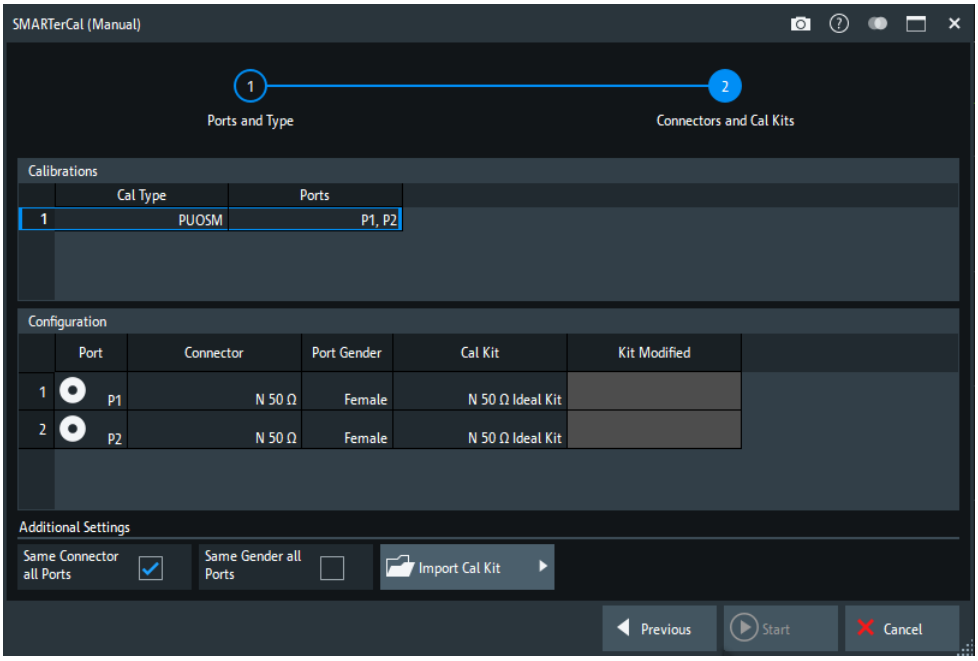
Remote command:
`[SENSe<Ch>:]CORRection:COLLect:METHod:DEFine`

Power Meter
Shows a list of all power meters that have been properly configured and are available for the scalar source power calibration of a single source port. The last configured power meter is selected by default. See "Configured Devices" on page 846.

Remote command:
`[SENSe:]CORRection:COLLect:PMETer:ID`

Step 2: Connectors and Cal Kits

Selects the connector type and gender for all ports and allows you to import a calibration kit.



The upper part of the panel shows the previously configured SMARTerCal(s). Select the appropriate one to get access to the related port settings. For the individual calibrations, the functionality is the same as described for the single calibration version of the wizard (see "Step 2: Connectors and Cal Kits" on page 620).

Back
Go back to Step 2: Connectors and Cal Kits.

Start
Start Step 3: Calibration.

Step 3: Calibration

Allows you to acquire error correction data for every required port (pair) and calibration standard, plus power correction data from the power meter.

Same functionality as for the single calibration version of the wizard (see "Step 3: Calibration" on page 587).

- The order of the system error correction and power calibration sweeps is arbitrary, but make sure that you always connect the proper equipment.

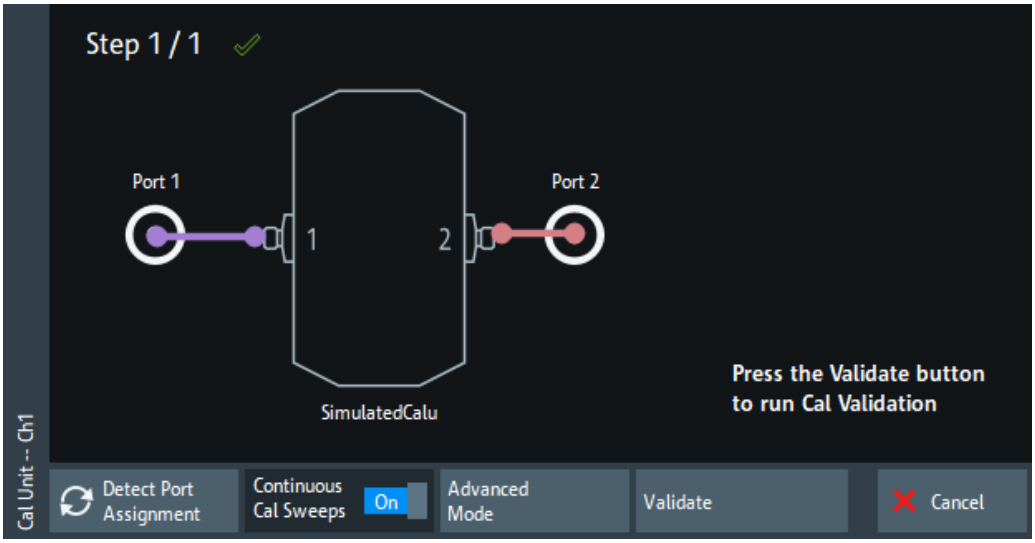
The possibility to "share" measurements between the calibrations and the "Reduced Through" logic help to keep the number of measurements as small as possible (see Section 4.5.1.11, "Full n-Port calibration with reduced number of Through connections", on page 166).

5.11.1.12 Cal Validation dock widget

The controls on the "Cal Validation" dock widget allow you to set up and run a Calibration validation in basic or advanced mode.

Basic mode

In basic mode, the validation runs automatically, with autodetected ports and the settings specified in Advanced mode (default settings after a system [Preset]).



Use "Detect Port Assignment" to detect the current port assignment (after reconnecting the calibration unit).

Manual port assignment is not supported.

Continuous Cal Sweeps
See "Continuous Cal Sweeps" on page 814.

Validate
Runs the validation for the current port assignment. The validation proceeds through the connected ports cal unit ports p ∈ {1, ..., 4} in ascending order, measuring S_{pp} for the Open, Short and Match standard.

- "PASS": all measured reflection parameters are within the specified Validation Limits, compared to the reflection parameters of the calu's characterization..
- "FAIL" otherwise.
  [In this case, the validation stops at the first failed reflection measurement.]

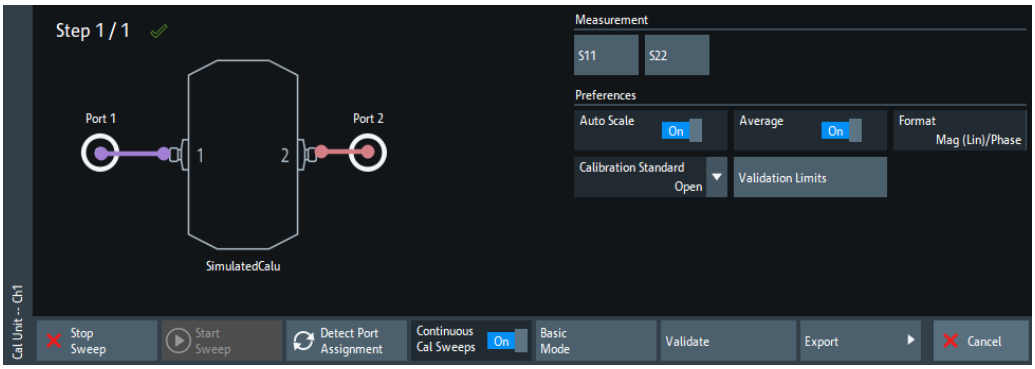
Remote command:
`CALCulate:CALValidate:RUN`
`CALCulate:CALValidate:RUN:RESult?`

Advanced Mode
Switches to Advanced mode.

Remote command:
`CALCulate:CALValidate:MODE` on page 918

Advanced mode

In advanced mode you can configure the validation settings and run or rerun particular measurements (Measurement, "Calibration Standard" on page 632).



Use "Detect Port Assignment" to detect the current port assignment (after reconnecting the calibration unit).

Measurement
Selects the measurement to be performed in advanced mode and runs it.

Calibration Standard
Selects the calibration standard to be measured in advanced mode.

Remote command:
`CALCulate:CALValidate:STANdard`

Validation Limits
The "Limits..." button opens the "Cal Validation Limits" dialog that allows you to specify the maximum tolerable deviations from the corresponding S-parameters of the calu's characterization.



Figure 5-65: Validation limits for Mag (dB)/Phase format

Depending on the selected Format, different limits can be set.

- "Mag (Lin)/Phase": lower and upper limits for magnitude (linear) and phase; defaults are ±52 mU for magnitude and ±8° for phase deviations.
- "Mag (db)/Phase": lower and upper limits for magnitude (logarithmic) and phase; defaults are ±0.2 dB for magnitude and ±8° for phase deviations.

Note that phase limits are only checked for the high reflection standards Open and Short, and **not** for Match.

Remote command:

```
CALCulate:CALValidate:RESPonse:MAGNitude:LOWer
CALCulate:CALValidate:RESPonse:MAGNitude:UPPer
CALCulate:CALValidate:RESPonse:LMAGnitude:LOWer
CALCulate:CALValidate:RESPonse:LMAGnitude:UPPer
CALCulate:CALValidate:RESPonse:PHASe:LOWer
CALCulate:CALValidate:RESPonse:PHASe:UPPer
CALCulate:CALValidate:RESPonse:PRESet
```

Preferences

Allows you to configure the behavior of the validation diagrams and traces.

Disabling "Auto Scale" prevents the traces from scaling as the measurement and/or standard changes. Disabling "Average On" turns off averaging.

"Format" selects the trace format for the limit check (see "Validation Limits" on page 632).

Since FW V3.70, the "Imag/Real" format is no longer supported.

Remote command:

```
CALCulate:CALValidate:AVERage[:STATe]
CALCulate:CALValidate:FORMat
```

Basic Mode

Switches to Basic mode

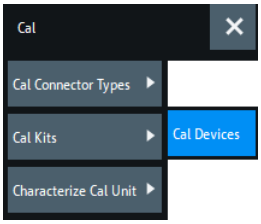
5.11.2 Cal Devices tab

Provides access to all functions for calibration kit management and cal unit characterization.

Background information

Refer to the following sections:

- Section 4.5.2, "Calibration standards and calibration kits", on page 168
- Section 4.5.5.3, "Characterization of calibration units", on page 179



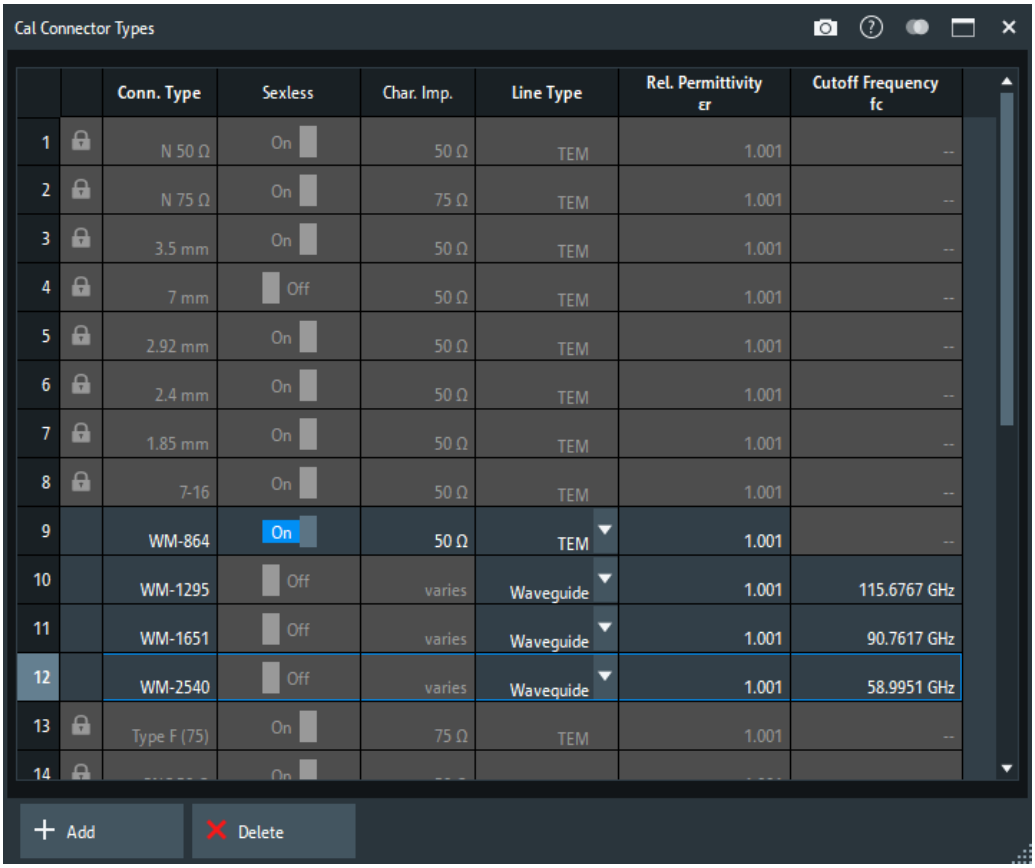
The buttons in the "Cal Devices" tab open the following dialogs:

- **"Cal Connector Types"**: See Section 5.11.2.1, "Cal Connector Types dialog", on page 634
- **"Cal Kits":** See Section 5.11.2.2, "Calibration Kits dialog", on page 637
- **"Characterize Cal Unit":** See Section 5.11.2.3, "Characterize Cal Unit dialog", on page 643

5.11.2.1 Cal Connector Types dialog

The "Cal Connector Types" dialog displays and modifies the list of available connector types. Cal connector types must be selected in accordance with the connectors of the measured DUT.

Access: Channel – [Cal] > "Cal Connector Types"



The list shows the available connector types with their name ("Conn. Type") and polarity ("Sexless"). The remaining columns in the list are described below.

Storing connector type settings

Calibration kits and connector types are global resources; the parameters are stored independently and available for all recall sets. The connector type settings are always stored together with the associated calibration kit parameters. The Calibration Kits dialog provides buttons to export and import cal kit and connector settings.

After assigning a calibration kit to a user-defined connector type, you can still change its name, offset model and reference impedance. If you switch between sexed and sexless, all kits assigned to the connector type are deleted.

Char. Imp.

The characteristic impedance or reference impedance ("Char. Imp.") Z_0 for the connectors is a critical value that has an impact on various parameter conversions. Z_0 enters into:

- The calculation of the S-parameters for the calibration standards associated with the connector type, if they are derived from a circuit model (see "View / Modify Cal Kit Standards dialog" on page 641).
- The calculation of the (default) reference impedances for balanced ports (see "Reference Impedance tab" on page 300).

- The calculation of impedance and admittance parameters (see Section 4.3.3, "Impedance parameters", on page 121 and Section 4.3.4, "Admittance parameters", on page 125).

Remote command:
`[SENSe:]CORRection:CONNection`

Line Type / Rel. Permittivity εr / Cutoff Freq. fc
"Line Type" describes the wave propagation mode (offset model) in the transmission lines of the standards associated with the connector type.

- If the calibration kit standards contain lines with transverse electric propagation mode (TEM, e.g. coax cables), then the "Rel. Permittivity εr" of the dielectric can be defined. The default permittivity is the value for air. TEM-type lines have no cutoff frequency.
- If the calibration kit standards contain waveguides, then the lowest frequency where a wave propagation is possible ("Cutoff Freq. fc") can be defined. The default cutoff frequency if 0 Hz (propagation at all frequencies). No relative permittivity is needed for waveguides.

Note: The impedance for waveguides is frequency-dependent. If a waveguide line type is selected, various dialogs indicate "varies" instead of a definite impedance value.

Impact of line type parameters

The line type parameters are used for the calculation of the S-parameters for the calibration standards associated with the connector type, if they are derived from a circuit model (see "View / Modify Cal Kit Standards dialog" on page 641).

- For TEM-type lines, the relative permittivity ε_r is needed for the conversion of a ZVR-type "Loss" (in units of dB/sqrt(GHz)) into a Keysight-type "Offset Loss" (in units of GΩ/s) and vice versa (see "View / Modify Cal Kit Standards dialog" on page 641). The "Electrical Length" and "Delay" values in the View / Modify Cal Kit Standards dialog are directly entered and therefore independent of ε_r.
- For waveguides, the low frequency cutoff frequency f_c is important because no wave propagation is possible at frequencies below f_c. If a standard is measured to acquire calibration data, the analyzer checks the low frequency cutoff. If the start frequency of the sweep range is below f_c , an error message is generated.

The offset model parameters are not used except in the context of calibration. The offset parameter definitions are based on independent ε_r values; see Section 5.13.2, "Offset tab", on page 737.

Remote command:
`[SENSe:]CORRection:CONNection`

Add / Delete
Adds or deletes a user-defined connector type. The parameters of a user-defined connector type can be modified in the table.

Note: Deleted/Missing Connector Types.

- Deleting a connector type also deletes all calibration or adapter kits assigned to it.
- Deleting a connector type that is used by a loaded recall set resets the affected ports to the instrument's connector type and gender.
- A setup can only be loaded if all its connector types (identified by their names) are configured at the target instrument.

Remote command:
[SENSe:]CORRection:CONNection
[SENSe:]CORRection:CONNection:CATalog?
[SENSe<Ch>:]CORRection:CONNection:DELete

5.11.2.2 Calibration Kits dialog

The "Calibration Kits" dialog shows the available calibration kits for the different connector types. It is also used for cal kit and cal kit file management.

Access: Channel – [Cal] > "Cal Devices" > "Cal Kits"

Related information

Refer to the following sections:

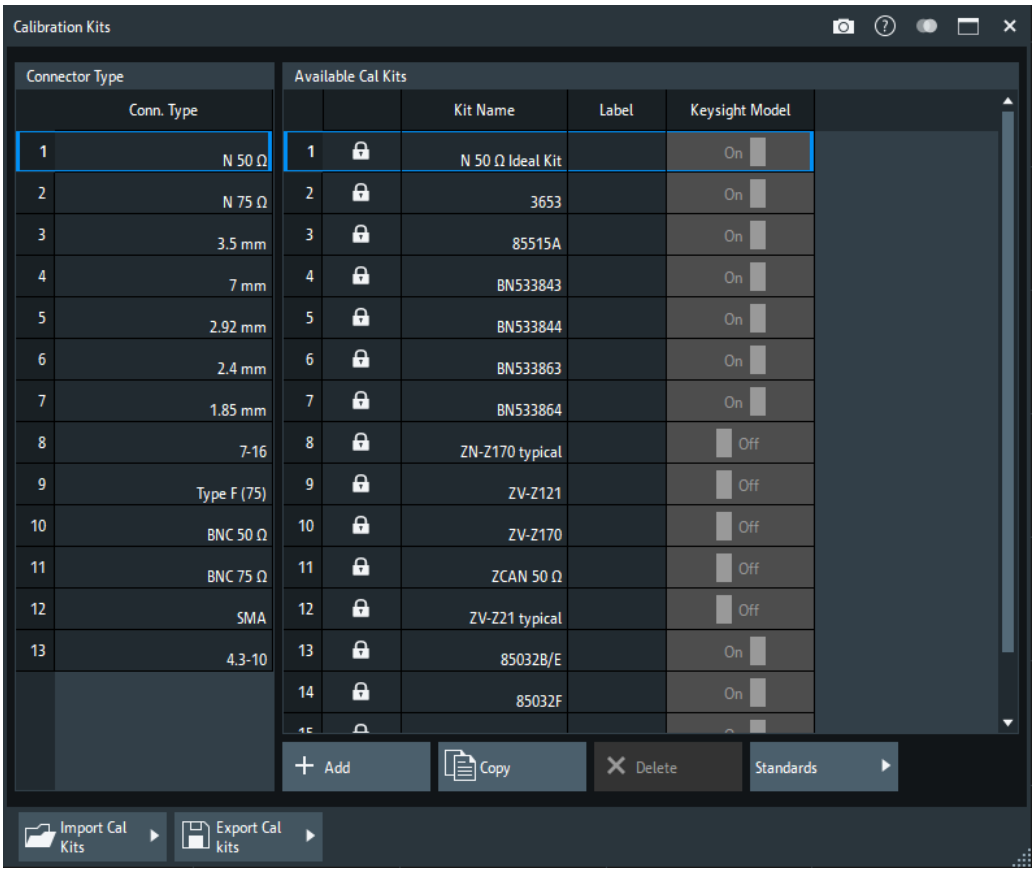
- Section 4.5.2, "Calibration standards and calibration kits", on page 168
- Section 4.5.2.4, "Cal kit Files", on page 173
- Section 4.5.2.2, "Cal kit parameter types", on page 171
- Section 5.11.2.1, "Cal Connector Types dialog", on page 634

The contents of the "Available Cal Kits" table vary, depending on the selected "Connector Type". The table can also contain kits with ideal or typical parameter values; see Cal kit parameter types. The "Keysight Model" is an optional scheme to characterize the offset parameters of the standards; see "Offset Parameters" on page 642.

Cal kit labels

Assigning a "Label" to user-defined calibration kits is optional. However, the label is displayed in many dialogs and can provide useful information about the kit, e.g. its serial number. It is even possible to assign several calibration kits with the same name, distinguished by their label, to a common connector type. See also Section 7.3.14.5, "[SENSe:]CORRection:CKIT... with labels", on page 1324.

Controls in the Calibration Kits dialog



Connector Type
The "Connector Type" table displays the available cal kit connector types. Select a row in this table to get the list of Available Cal Kits.

Remote command:
`[SENSe:]CORRection:CONNection:CATalog?`

Available Cal Kits
Displays the cal kits for the selected Connector Type.

If option K50 "Measurement Uncertainty Analysis" is installed, the "KitModified" column is replaced by an "Uncertainty" column, indicating whether the kits contains uncertainty information for all its calibration standards.

Remote command:
`[SENSe:]CORRection:CKIT:CATalog?`
`[SENSe:]CORRection:CKIT:<ConnType>:SELect`
`[SENSe:]CORRection:CKIT:<ConnType>:LSELect`

Add / Copy / Delete / Standards
The buttons in the right part of the dialog are used to manage calibration kits:
- "Add" creates a cal kit file for the selected connector type.
- "Copy" creates a cal kit file based on the contents of an existing cal kit file.
- "Delete" deletes an imported or user-defined cal kit file.

- "Standards" opens the "Kit Standards" dialog, which shows the contents of the cal kit file. For user-defined or imported kits, you can modify the contents. See "Kit Standards dialog" on page 639.

Remote command:
The following commands create and modify calibration kits:
```
[SENSe:]CORRection:CKIT:DMODe
[SENSe:]CORRection:CKIT:ADD
[SENSe:]CORRection:CKIT:COPY
[SENSe:]CORRection:CKIT:DELete
```
Query connector types and calibration kits:
```
[SENSe:]CORRection:CONNection:CATalog?
[SENSe:]CORRection:CKIT:CATalog?
```

Import Cal Kit / Export Cal Kit
The buttons below the "Connector Type" list are used to store cal kit data to a file and to reload previously stored cal kit files. By default, calibration kit files are stored in the `C:\Users\Public\Documents\Rohde-Schwarz\Vna\Calibration` directory; see Section 4.5.2.4, "Cal kit Files", on page 173.

Remote command:
```
MMEMory:LOAD:CKIT
MMEMory:STORe:CKIT
```

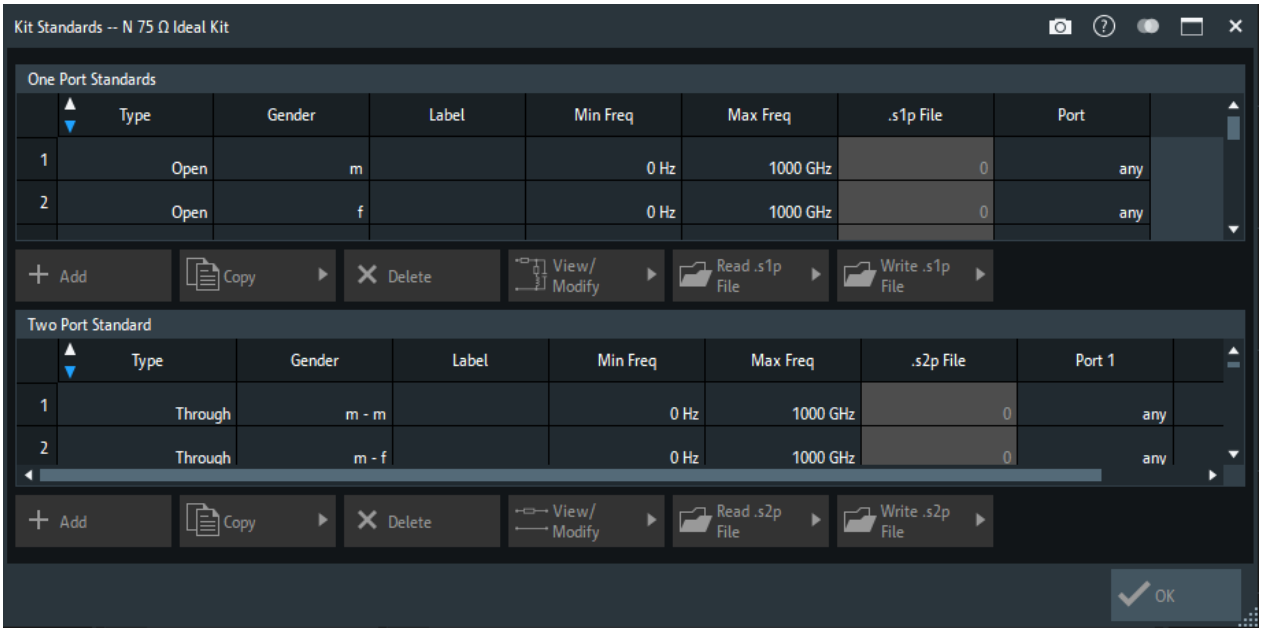
Kit Standards dialog

The "Kit Standards" dialog shows the calibration standards in a selected calibration kit. It is also used to modify the contents of a user-defined kit.

Related information
Refer to the following sections:

- Section 4.5.2.4, "Cal kit Files", on page 173
- Section 5.11.2.2, "Calibration Kits dialog", on page 637
- Section 4.5.2.1, "Calibration standard types", on page 169

Access: Calibration Kits dialog > "Standards"



One port and two port standards are listed in two separate tables. Most of the buttons on the right side are available only if the "Kit Standards" dialog was opened for a user-defined calibration kit.

One Port Standards / Two Port Standards

The standard tables contain the following information:

- "Type" and "Port Gender" describe the calibration standard type; for an overview see Section 4.5.2.1, "Calibration standard types", on page 169.
- "Label" is a user-defined name of the standard. The label can help you identify a standard or distinguish different standards with similar parameters.
- "Min Freq" and "Max Freq" define the rated frequency range of the standard. During calibration, the analyzer checks whether the sweep range is within the validity range of all measured standards and possibly generates a warning.
- ".s1p File" and ".s2p File" define whether the characteristics of the standard are described by a Touchstone file rather than by a circuit model from which the R&S ZNB can calculate the S-parameters. See "Read .s1p File / Read .s2p File /" on page 641 and "View / Modify Cal Kit Standards dialog" on page 641.
- "Port" defines whether the standard can be connected to any analyzer port or to just one port (for one-port standards) or a pair of ports (for two-port standards). Standards with unrestricted port assignment ("any") are stored with their gender. When a connector type and calibration kit are selected for the calibration, the analyzer checks whether the kit contains the required standard types and whether the standards have the right gender.
  Standards with restricted port assignment are always assumed to have the gender that is appropriate for the calibrated port. The **port assignment** is stored in the calibration kit file, instead of the gender. During the calibration, the analyzer checks whether the cal kit contains the necessary standard types for the required ports.

Remote command:

```
[SENSe:]CORRection:CKIT:<StandardType>
[SENSe:]CORRection:CKIT:<ConnType>:SELect
```

Add / Copy / Delete / View / Modify
The buttons in the right part of the dialog are used to manage standards:

- "Add" adds a new standard to the calibration kit. The properties of the standard can be edited in the table.
- "Copy" creates a standard based on the properties of an existing standard.
- "Delete" deletes the selected standard.
- "View / Modify" opens the "View / Modify Cal Kit Standards" dialog. This dialog shows the circuit model for the selected standard. For user-defined standard, you can modify the circuit model parameters. See "View / Modify Cal Kit Standards dialog" on page 641.

Remote command:
`[SENSe:]CORRection:CKIT:<StandardType>`
`[SENSe:]CORRection:CKIT:<StandardType>:WLABel`

Read .s1p File / Read .s2p File /
Opens a file selection dialog where you can select a Touchstone file containing the reflection or transmission S-parameters for the standard. The R&S ZNB uses the imported S-parameters rather than the circuit model to characterize the standard, if ".s1p File"/".s2p File" is enabled in the standard table. The appropriate file type (`*.s1p` for one-port standards and `*.s2p` for two-port standards) is selected automatically.

Remote command:
`MMEMory:LOAD:CKIT:SDATa`

Write .s1p File/Write .s2p File
This button is only enabled, if the selected 1-port/2-port standard is defined using an s1p/s2p file. It opens a dialog that allows you to export the respective s<n>p file.

This feature significantly facilitates the creation of custom cal kits.

Remote command:
n.a.

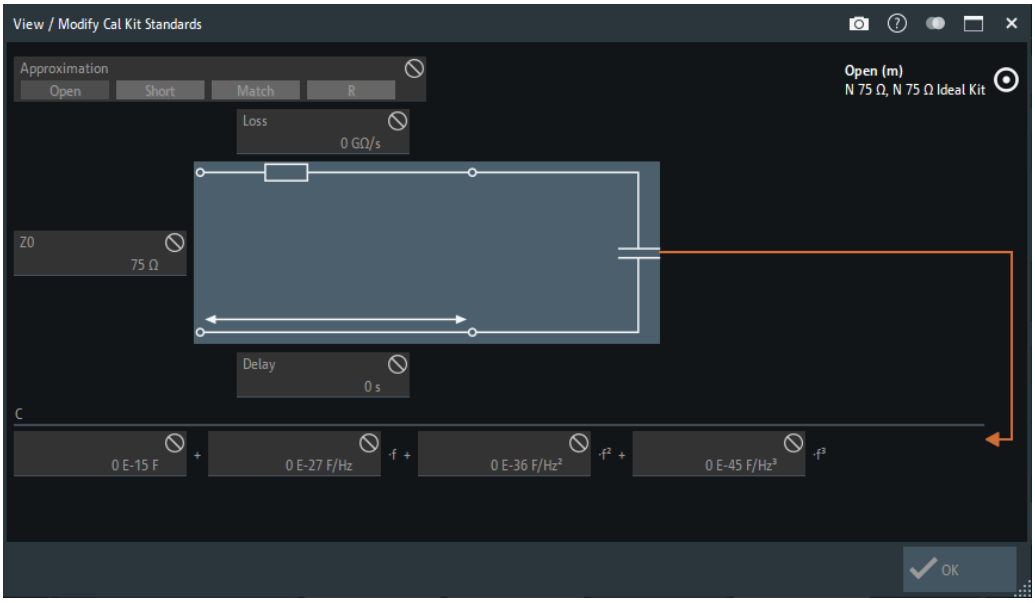
View / Modify Cal Kit Standards dialog

The "View / Modify Cal Kit Standards" dialog shows the circuit model for a selected calibration standard. It is also used to define or edit the circuit model (offset and load) parameters for a user-defined standard.

Related information
Refer to the following sections:

- Section 4.5.2.4, "Cal kit Files", on page 173
- "Kit Standards dialog" on page 639
- Section 4.5.2.1, "Calibration standard types", on page 169

Access: Kit Standards dialog > "View / Modify"



The diagram in the "View / Modify Cal Kit Standards" dialog depends on the standard type for which the dialog was opened. Moreover, it is possible to modify the circuit model using the buttons in the upper right of the dialog.

Offset Parameters

The entries in the upper part of the "View / Modify Cal Kit Standards" dialog specify the offset parameters for the transmission lines of the selected calibration standard.

The offset parameters depend on whether the circuit model is defined as "Keysight Model" (see Section 5.11.2.1, "Cal Connector Types dialog", on page 634):

- In a "Keysight Model", a calibration standard is characterized by its "Delay" (in s), its characteristic impedance "Z0" (in Ω) and its "Offset Loss" (in GΩ/s).
- Otherwise the standard is characterized by the R&S ZVR-compatible parameters "Electrical Length" (in m), "Char. Imp." (in Ω) and "Loss" (in dB/sqrt(GHz)). The loss is zero and not editable as long as the electrical length is zero.

Both parameter sets are closely related. The "Electrical Length" is proportional to the "Delay"; "Z0" corresponds to the "Char. Imp.". Moreover the analyzer converts a Keysight-type "Offset Loss" into a R&S ZVR-type "Loss" and vice versa using the "Rel. Permittivity εr" for the selected connector type.

See also description of the offset parameters in Section 4.5.2.1, "Calibration standard types", on page 169.

Remote command:

`[SENSe:]CORRection:CKIT:<StandardType>`

Load Parameters

The entries in the lower part of the "View / Modify Cal Kit Standards" dialog specify the load parameters for a particular calibration standard describing its terminal impedance.

The circuit model for the load consists of capacitance C which is connected in parallel to an inductance L and a resistance R, both connected in series.

- R is the constant resistive contribution. It is possible to select:
  – "Open" for ∞ Ω (so that the inductance coefficients are irrelevant)

  - "Short" for 0 Ω
  - "Match" for the reference impedance of the current connector type
  - any resistance "R"
- The fringing capacitance C and the residual inductance L are both assumed to be frequency-dependent and approximated by the first four terms of the Taylor series around f = 0 Hz.

See also description of the load parameters for the different standard types in Section 4.5.2.1, "Calibration standard types", on page 169.

Remote command:
`[SENSe:]CORRection:CKIT:<StandardType>`

5.11.2.3 Characterize Cal Unit dialog

The "Characterize Cal Unit" dialog displays the properties of the connected cal units, provides control elements for characterization file management, and starts the characterization wizard.

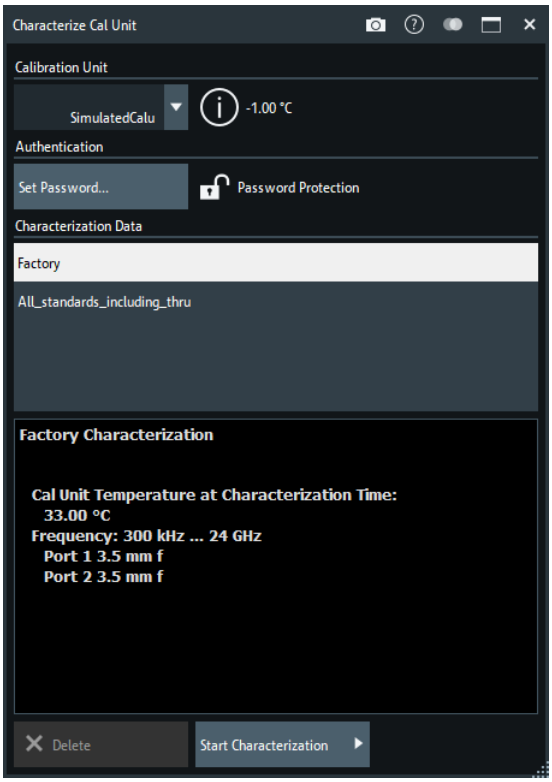
Access: Channel – [Cal] > "Cal Devices" > "Characterize Cal Unit"

Background information

Refer to Section 4.5.5.3, "Characterization of calibration units", on page 179.

A cal unit characterization can be performed in a frequency sweep. The "Characterize Cal Unit" dialog is unavailable while a "Power", "CW Mode", or "Time" sweep is active. The analyzer always uses a fixed source power of -10 dBm to acquire the characterization data.

Controls in the Characterize Cal Unit dialog



Calibration Unit
Displays the connected calibration units. The R&S ZNB auto-detects all calibration units which are connected to one of its USB ports. If several cal units are connected, one of them must be selected for characterization (active cal unit).

Remote command:
`SYSTem:COMMunicate:RDEVice:AKAL:ADDRess:ALL?`
`SYSTem:COMMunicate:RDEVice:AKAL:ADDRess`
`SYSTem:COMMunicate:RDEVice:AKAL:TEMPerature?`

Authentication
Allows you to set a password to protect the characterization dialog and the Characterization wizard from unauthorized access and operation. "Set Password" opens a dialog to enter the password and activate password protection at the next time the "Set Password" dialog is opened. Enter an empty string (no password) to deactivate password protection.

Tip: A password also blocks a switchover of the active characterization during calibration; see "Characterization" on page 568.

Remote command:
`[SENSe:]CORRection:COLLect:AUTO:CKIT:PASSword`

Characterization Data
Displays all characterizations which are stored on the active cal unit. The "Factory" characterization is available for all calibration units; it ensures an accurate calibration for all standard applications. Characterizations stored on an SD card (inserted at the cal unit) are prefixed with "SD:".

Tip: Characterizations stored on an SD card (inserted at the cal unit) are prefixed with "SD:".

The properties of the selected characterization are shown below the list. "Delete" deletes the selected characterization file; "Start Characterization" opens the Characterization wizard to create a characterization.

Remote command:
`SYSTem:COMMunicate:RDEVice:AKAL:CKIT:CATalog?`
`SYSTem:COMMunicate:RDEVice:AKAL:CKIT:STANdard:CATalog?`
`SYSTem:COMMunicate:RDEVice:AKAL:SDATa?`

Characterization wizard

The "Characterization" wizard guides you through the automatic characterization of a calibration unit.

Access: Characterization wizard > "Start Characterization"

The guided characterization consists of the following steps:

1. **"Characterization":** Select the characterized ports and cal unit standards to initiate the characterization sweeps.
2. **"Save Characterization Data":** Save the characterization data to the calibration unit.

Step 1: Characterization

Selects the calibration type and the characterized cal unit ports and initiates the necessary characterization sweeps.

Characterization procedure
To acquire accurate characterization data, the test setup must be properly calibrated before you start the characterization wizard. Use the calibration type that you wish to perform with your new cal unit characterization; see Section 4.5.5.3, "Characterization of calibration units", on page 179.

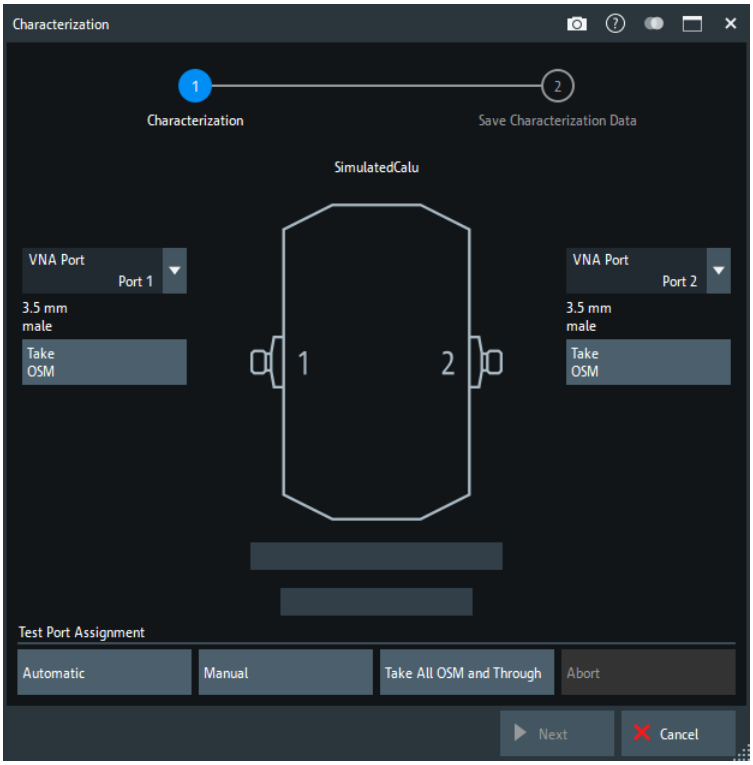


Figure 5-66: Characterization of standard calibration units



Figure 5-67: Characterization of inline calibration units

Test Port Assignment

Defines the assignment between test ports and cal unit ports. In the default "Manual" assignment, VNA ports and cal unit port numbers match. If you decide to use a different assignment, you can auto-detect the actual assignment ("Automatic") or select the analyzer port numbers manually. Auto-detection can fail, e.g., because of a high attenuation in the signal path.

Remote command:

```
[SENSe<Ch>:]CORRection:COLLect:AUTO
[SENSe<Ch>:]CORRection:COLLect:AUTO:PORTs
[SENSe:]CORRection:COLLect:AUTO:PORTs:CONNection?
[SENSe:]CORRection:COLLect:AUTO:CKIT:PORTs
```

Take OSM / Take All OSM and Through
"Take OSM" starts a calibration sweep for the related port. "Take All OSM and Through" initiates a series of calibration sweeps; the R&S ZNB acquires a full set of one-port and two-port data. The latter is required for the transmission normalizations and for a "One Path Two Ports" calibration; see "Dependency between calibration types and characterization data" on page 179.

Remote command:
`[SENSe<Ch>:]CORRection:COLLect:AUTO`
`[SENSe<Ch>:]CORRection:COLLect:AUTO:PORTs`
`[SENSe:]CORRection:COLLect:AUTO:PORTs:CONNection?`

Next
Proceeds to Step 2: Save Characterization Data . Next is available when the R&S ZNB has acquired characterization data for a single port.

Step 2: Save Characterization Data

Saves the characterization data to the calibration unit.

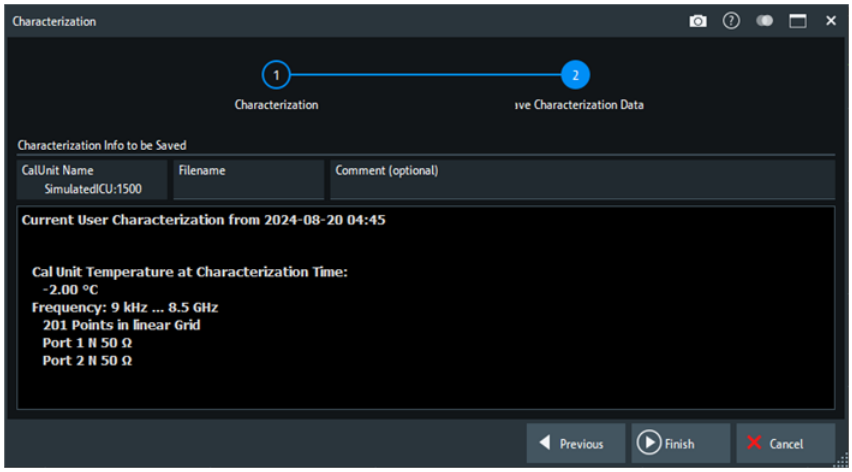


Figure 5-68: Save Characterization Data

Save File to SD Card
For all calibration units, characterization data can be saved to the calibration unit's internal flash memory. For some calibration units (e.g. the new models R&S ZN-Z5x and R&S ZN-Z15x), they can also be saved to an SD card inserted at the calibration unit. Enable this control to save the characterization data to the SD card.

Tip: If the characterized calibration unit does not have an SD card slot, the control is hidden. If the calibration unit has an SD card slot but the SD card is not accessible, the control is grayed out.

File name / Comment (Optional)
Selects a filename to reference the characterization data set in the "Characterize Cal Unit" and "Calibration Unit" dialogs and a comment, to be written into the characterization file. A filename is required before you can "Finish" the characterization wizard and store the data.

Remote command:
`[SENSe:]CORRection:COLLect:AUTO:CKIT`

5.11.3 Power Cal Settings tab

Gives access to all functions for power meter and power calibration data handling (transmission coefficients). Power calibration is a fully menu-guided process.

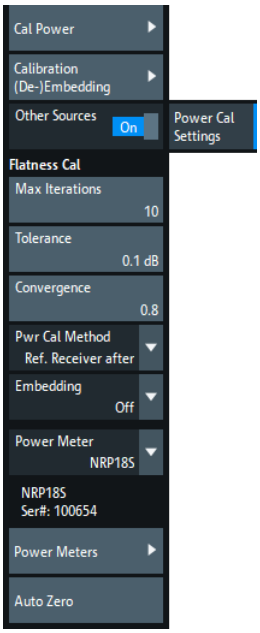
Efficient power calibration procedure

- For standard applications, open the "Start Cal" tab and select "Scalar Power Cal" – "Power Cal" to perform the necessary calibration sweeps with default power calibration settings. You do not need any of the buttons in the "Power Cal Settings" tab.
- Select "Cal Power" if you use an amplifier between the source port and the DUT.
- Select "Calibration (De-)Embedding" if you want to modify the power calibration procedure.

Background information

Refer to Section 4.5.6, "Scalar power calibration", on page 187.

5.11.3.1 Controls on the Power Cal Settings tab



Cal Power

Opens the Cal Power Config dialog.

Calibration (De-)Embedding
Opens the Calibration (De-)Embedding dialog.

Other Sources
Allows you to switch off all sources except the calibrated source during calibration. This is recommended if the measurement involves a combination of different signals.

Remote command:
`SOURce<Ch>:POWer:CORRection:OSOurces[:STATe]`

Flatness Cal – Max Iterations
Sets a limit for the number of calibration sweeps. See also "Calibration procedure" on page 189.

Remote command:
`SOURce:POWer:CORRection:NREadings`
`SOURce:POWer:CORRection:COLLect:AVERage[:COUNt]`

Flatness Cal – Tolerance
Defines the maximum deviation of the measured power from the cal power. The calibration procedure is stopped if "Max Iterations" is reached or if the measured power is within the "Tolerance".

Remote command:
`SOURce:POWer:CORRection:COLLect:AVERage:NTOLerance`

Flatness Cal – Convergence
Modifies the amount of power correction for each of the flatness calibration sweeps. The power correction in each sweep, as controlled by the calibrated reference receiver (a-wave receiver), is multiplied by the selected convergence factor. With a convergence factor larger (smaller) than 1, the source power correction after each flatness calibration step is larger (smaller) than the measured deviation from the desired power.

For analyzer test ports, a convergence factor 1 is appropriate. Convergence factors different from 1 may be indicated for external generator ports that show a nonlinear behavior. In general, it is recommendable to start the calibration with a convergence factor 1 and choose smaller values (0.8 ... 0.4) in case that the iteration fails. Inappropriate convergence factors can slow down the flatness calibration or even prevent convergence.

Remote command:
`SOURce:POWer:CORRection:COLLect:CFACtor`

Flatness Cal – Pwr Cal Method
Defines how power calibrations are performed.

"Ref. Receiver after"	Use a power meter for the calibration of the reference receiver, and the calibrated reference receiver for the flatness calibration.
"Power Meter only"	Use a power meter for the calibration of the reference receiver *and* the flatness calibration.

"Ref. Receiver only" Perform a flatness calibration using the reference receiver. Do not recalibrate the reference receiver; use the best available reference receiver calibration instead.

For external generators, a power meter must be used.

"Ref. Receiver after" is the default behavior.

Remote command:
`SOURce:POWer:CORRection:COLLect:METHod`

Embedding
Defines whether the source flatness cal uses the configured offset de-/embedding to get the power values at the DUT (instead of the calibrated reference plane).

This setting applies to all ports and channels. Default is "Off" (no offset or de-/embedding used).

Remote command:
`SOURce:POWer:CORRection:COLLect:VNETworks`

Power Meter
Shows a list of all power meters that have been properly configured and are available for the power calibration of a source port. The last configured power meter is selected by default. See "Configured Devices" on page 846.

Remote command:
`SOURce:POWer:CORRection:PMETer:ID`

Power Meters
Opens the External Power Meters dialog.

Auto Zero
Initiates an automatic zeroing procedure of the selected power meter.

The power meter must be disconnected from the RF power; see Section 4.7.28.1, "Zeroing", on page 268. A message indicates that zeroing is finished.

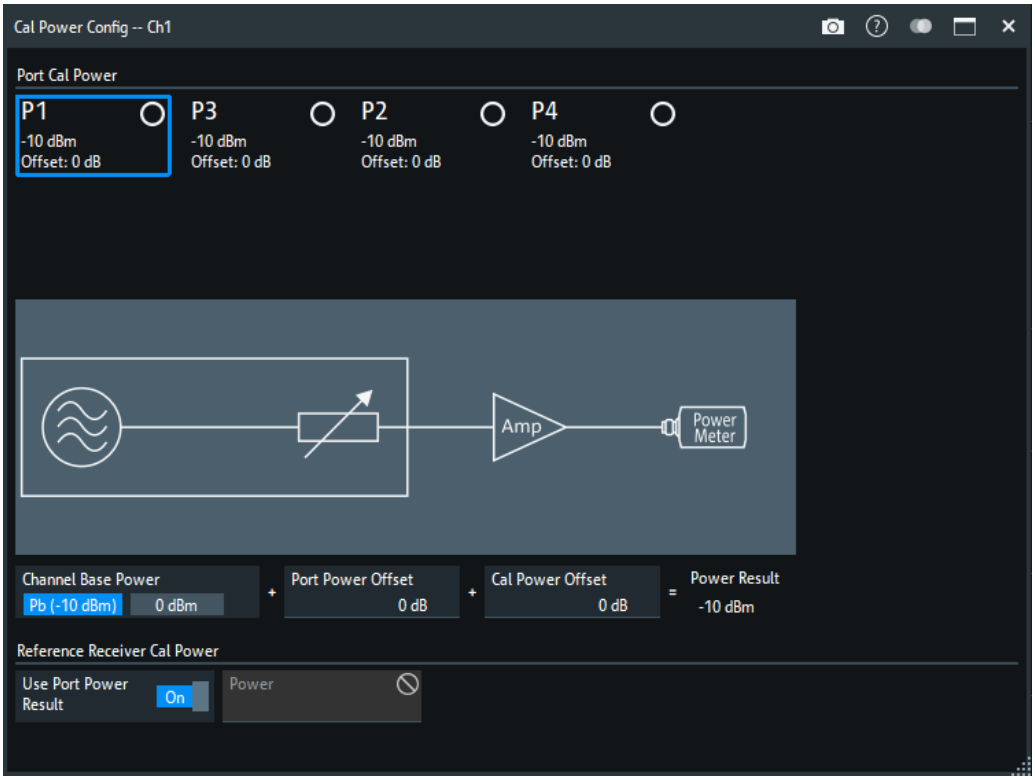
Remote command:
`SYSTem:COMMunicate:RDEVice:PMETer<Pmtr>:AZERo`

5.11.3.2 Cal Power Config dialog

The "Cal Power Config" dialog adjusts the target power for the power calibration (cal power) and defines the target power for the reference receiver calibration. These settings are particularly important for test setups involving external attenuators or amplifiers.

Access: Channel – [Cal] > "Pwr Cal Settings" > "Cal Power..."

Related information
The settings in the "Cal Power Config" dialog are also used to define port-specific source powers in arbitrary mode; refer to Arbitrary Power tab.



The dialog shows all source ports of the network analyzer. The diagram visualizes the calibration setup.

Port Cal Power
Select a source port to define its port-specific power levels for source power calibrations.

Configured external generators ("G1" ...) are included in the source port overview.

Channel Base Power ← Port Cal Power
Allows you to select between the channel base power P_b and a fixed value of 0 dBm.

Remote command:
`SOURce<Ch>:POWer<PhyPt>[:LEVel][:IMMediate]:OFFSet`

Port Power Offset ← Port Cal Power
Defines a port-specific offset to the Channel Base Power The actual output power at the port is equal to the "Channel Base Power" plus the "Port Power Offset".

If P_b is selected as "Channel Base Power", then for a power sweep the actual port power varies across the sweep. Otherwise the port power is constant.

Remote command:
`SOURce<Ch>:POWer<PhyPt>[:LEVel][:IMMediate]:OFFSet`
Generator: `SOURce<Ch>:POWer<PhyPt>:GENerator<Gen>:OFFSet`

Cal Power Offset ← Port Cal Power
For power calibrations only: Specifies a gain (positive values) or an attenuation (negative values) in the signal path between the source port and the calibrated reference plane. With a "Cal Power Offset" of n dB, the target power at the reference plane (cal power) is equal to the actual output power at the port plus n dB. The "Cal Power Offset" has no impact on the source power.

Example: Use of an amplifier in the signal path

Assume that a DUT requires a constant input power of +25 dBm, and that the measurement path contains an amplifier with a 30 dB gain. After a reset of the analyzer, the channel power P_b is -10 dBm. Select a "Port Power Offset" of +5 dB at the calibrated source port and a "Cal Power Offset" of +30 dB. Then the source power calibration ensures that the constant input power of +25 dBm is maintained across the entire sweep range. The actual output power of the analyzer is -5 dBm.

Notice that a power calibration with an appropriate "Cal Power Offset" can prevent excess input levels at the DUT.

Remote command:
`SOURce<Ch>:POWer<PhyPt>:CORRection:LEVel:OFFSet`
Generator: `SOURce<Ch>:POWer<PhyPt>:CORRection:GENerator<Gen>:LEVel:OFFSet`

Reference Receiver Cal Power
Defines the source power settings the R&S ZNB uses to perform the first calibration sweep of the source power calibration.

In this first sweep, the power meter reading is used to calibrate the reference receiver of the calibrated port. The following calibration sweeps are based solely on the reference receiver (see "Calibration procedure" on page 190).

For R&S ZNB instruments equipped with SYNTH9 synthesizers, the "Reference Receiver Cal Power" is set to the resulting Port Cal Power by default. For all other R&S ZNB, it defaults to a fixed "Power" value of -10 dBm. In any case, you can either use the "Port Cal Power" or a fixed power value. Note, however, that the accuracy of the source power calibration depends on the power meter's measurement accuracy. Therefore it is advantageous to select a "Reference Receiver Cal Power" at which the power meter provides maximum accuracy.

The "Reference Receiver Cal Power" is also used for the power calibration step in a SMARTerCal; see Section 4.5.7, "SMARTerCal", on page 193.

Note: Risk of damage due to high power settings. If an external device (e.g. an amplifier) is connected between the calibrated test port and the power meter, ensure that the "Reference Receiver Cal Power" does not exceed its maximum input power.

Remote command:
`SOURce:POWer:CORRection:PSELect`
`SOURce:POWer:CORRection:PPOWer`

5.11.3.3 Calibration (De-)Embedding dialog

Sometimes the power meter cannot be connected at the system error correction reference plane directly. For example, if an adapter is required at the power meter input.

The firmware models the following deviations:

- A two-port connected to the DUT which is not present during power calibration (e.g. an on-wafer probe)
- A two-port connected to the power meter during the power calibration (e.g. an adapter)

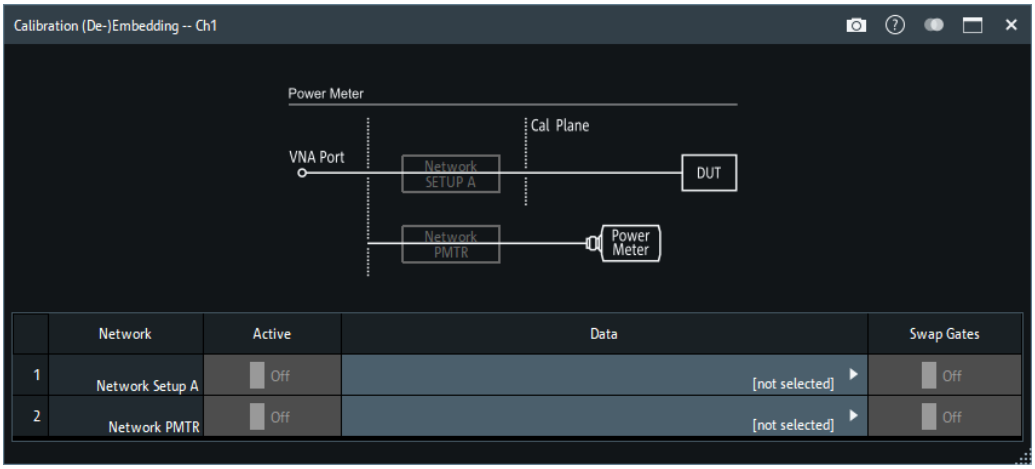
The "Calibration (De-)Embedding" dialog allows you to modify the results of a scalar power calibration to account for these two-ports.

Access: Channel – [Cal] > "Power Cal Settings" > "Calibration (De-)Embedding"

Background information

Refer to Section 4.5.6.5, "Extended test setups", on page 192.

Controls in the Calibration (De-)Embedding dialog



Active

Activates the respective 2-port network, defined in the Data column.

Remote command:

`SOURce:POWer:CORRection:TCOefficient[:STATe]`

Data

These cells open the Two Port Configuration dialog that allows you to define the respective two-port network.

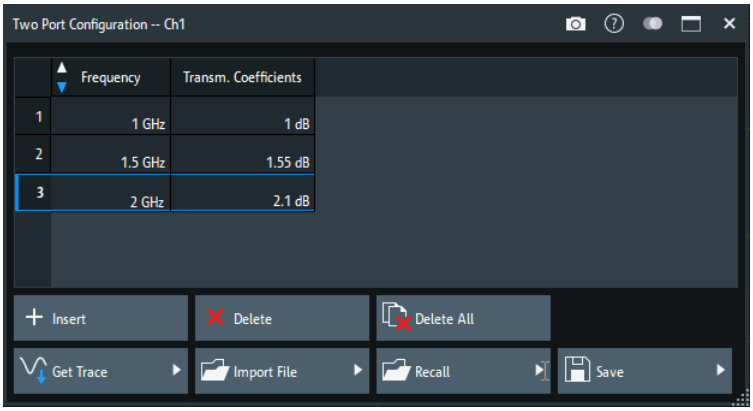
- "Network Setup A": A two-port between VNA and DUT (during measurement)
- "Network PMTR": A two-port between VNA and power meter (during power calibration)

See Section 4.5.6.5, "Extended test setups", on page 192.

Two Port Configuration dialog

The "Two Port Configuration" dialog defines the transmission characteristics of an additional two-port in the calibrated frequency range.

Access: Calibration (De-)Embedding dialog > "Data"



Frequency / Transm. Coefficients: Insert, Delete, Delete All
The required two-port information is a list of transmission coefficients at different frequency values (power loss list). The buttons in the dialog provide different ways of creating and modifying the list. Use "Insert", "Delete", "Delete All" to edit the list manually.

In a power, time or CW mode sweep, one point at the fixed CW frequency is sufficient. In a frequency sweep, it is possible to enter several coefficients to account for a frequency-dependent attenuation. Transmission coefficients are interpolated between the frequency points and extrapolated, if necessary.

If no transmission coefficient is defined at all, the VNA assumes a 0 dB attenuation across the entire frequency range. This assumption is equivalent to an ideal through connection or selecting "No Coefficients" in the Calibration (De-)Embedding dialog.

Remote command:
`SOURce:POWer:CORRection:TCOefficient:INSert<ListNo>`
`SOURce:POWer:CORRection:TCOefficient:DEFine<ListNo>`
`SOURce:POWer:CORRection:TCOefficient:COUNt?`
`SOURce:POWer:CORRection:TCOefficient:DELete<ListNo>[:DUMMy]`
`SOURce:POWer:CORRection:TCOefficient:DELete:ALL`

Get Trace
Opens a selection box containing all traces in the active recall set. The "dB Mag" values of the selected trace are used to define the transmission coefficients. Notice that if you combine different channels with different sweep points, the analyzer possibly has to interpolate or extrapolate the transmission coefficients.

Remote command:
`SOURce:POWer:CORRection:TCOefficient:FEED`

Import File
Imports the transmission coefficients from a trace file. The imported file must be either in Touchstone (`*.s<n>p`) or in `*.csv` format; see also Section 4.4.2, "Trace files", on page 143.

Remote command:
MMEMory:LOAD:CORRection:TCOefficient<Ch>

Recall / Save
You can save the displayed power loss list to a power meter correction list file with extension (*.pmcl) and reload it in later sessions.

Remote command:
MMEMory:LOAD:CORRection:TCOefficient<Ch>
MMEMory:STORe:CORRection:TCOefficient<Ch>

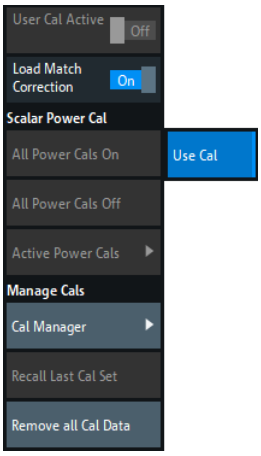
5.11.4 Use Cal tab

Provides access to functions for activating, deactivating and managing calibrations.

Background information
Refer to the following sections:

- Section 4.5, "Calibration", on page 154

5.11.4.1 Controls on the Use Cal tab



User Cal Active
Activates or deactivates the system error correction in the active channel. "User Cal Active" is available only if a valid system error correction is available for the active channel; see "Channel State" in Section 5.11.4.3, "Calibration Manager dialog", on page 657.

Note: A label "Cal Off" appears behind the trace list if the system error correction is switched off; see also Section 4.5.4, "Calibration state labels", on page 175. The calibration status of each channel and trace appears in the setup information ("Setup" > "Info" > "Setup").

Remote command:
[SENSe<Ch>:]CORRection[:STATe]

Load Match Correction
Load match correction prevents a possible impairment of the transmission S-parameter measurements due to the influence of the waves reflected at the receive (load) ports. It is performed as follows:

- The correction is based on the system error corrected a- and b-waves of the source and load ports. An appropriate calibration is a prerequisite for the load match correction; see Section 4.7.3.1, "Calibration options", on page 234.
- A reverse sweep is automatically performed during the measurement.
- The correction is calculated under the assumption that the reverse transmission factor is zero (e.g. S_{12}=0 if S_{21} is measured).

Load match correction can provide a significant improvement of the transmission S-parameter measurements if the load ports are poorly matched. With sufficiently matched load ports, you can disable the correction to gain speed.

Note: Disable the load match correction if your test setup or DUT is not suited for reverse sweeps or if you want to gain speed.

Remote command:
`[SENSe<Ch>:]FREQuency:CONVersion:GAIN:LMCorrection`

Scalar Power Cal – All Power Cals On / All Power Cals Off
Activates or deactivates all scalar power calibrations in the active channel. "All Power Cals On" is only available if a valid power calibration is available for the active channel, but not active; see "Channel State" in Section 5.11.4.3, "Calibration Manager dialog", on page 657.

Note: A label "PCal Off" appears behind the trace list of a wave quantity or a ratio if the power calibration is switched off; see also Section 4.5.6.4, "Power calibration labels", on page 191.
The calibration status of each channel and trace appears in the setup information (see "Setup tab" on page 828).

Remote command:
`[SENSe<Ch>:]CORRection:PCAL`

Active Power Cals
Opens the Active Power Cals dialog

Manage Cals – Recall Last Cal Set
Loads and activates the recall set for which the last calibration was performed. If the last calibrated setup is already active, nothing is changed. The calibrated setups are automatically stored in the `C:\Users\Public\Documents\Rohde-Schwarz\Vna\Calibration\RecallSets` directory. A message box pops up if the directory is empty, e.g. because no calibration was performed yet.

Remote command:
n/a

Manage Cals – Cal Manager
Opens the Calibration Manager dialog

Manage Cals – Remove all Cal Data
This button cleans the active channel's existing calibration data and also removes links to the calibration pool.

Remote command:
`[SENSe<Ch>:]CORRection:DELete`

5.11.4.2 Active Power Cals dialog

The "Active Power Cals" dialog shows the power calibrations that are compatible with the active channel configuration. It allows you to enable or disable the available power calibrations.

Access: Channel – [Cal] > "Use Cal" > "Active Power Cals"



All possible power calibrations can be performed from the Power Cal wizard.

Port Overview
Shows all source ports together with their possible power calibrations. Unavailable calibrations (i.e. calibrations that have not been performed yet) are grayed out.

If external generators are configured, they appear as additional source ports G1 ... Gm in the port overview. A source power calibration is available at each generator port.

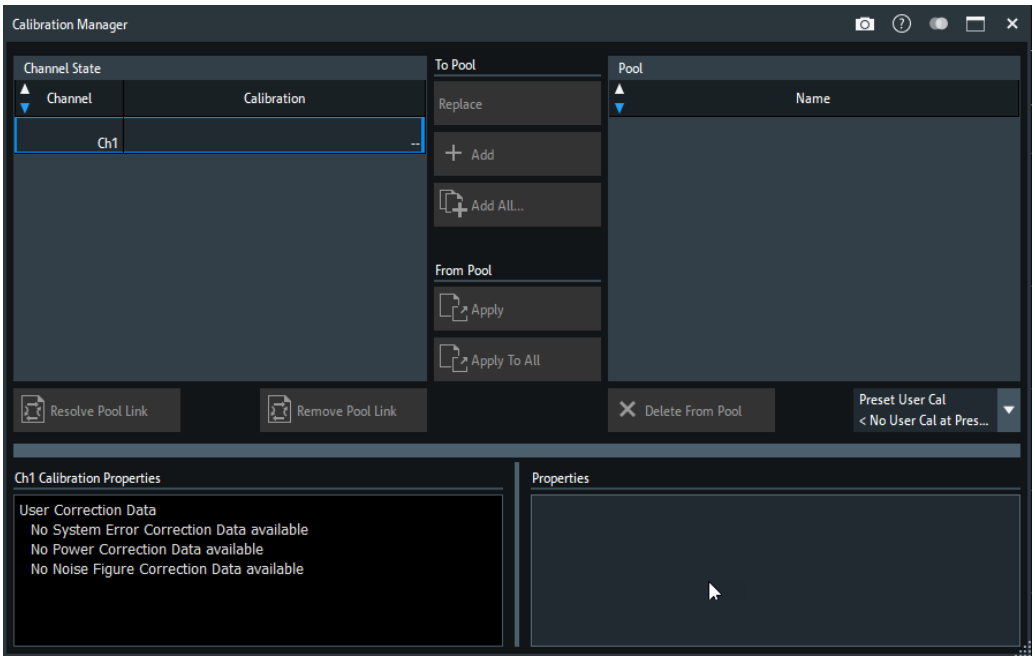
Remote command:
`SOURce<Ch>:POWer<PhyPt>:CORRection:STATe`
`SOURce<Ch>:POWer:CORRection:GENerator<Gen>[:STATe]`
`[SENSe<Ch>:]CORRection:POWer<PhyPt>[:STATe]`
`[SENSe<Ch>:]CORRection:PCAL`

5.11.4.3 Calibration Manager dialog

The "Calibration Manager" dialog stores user correction data to the cal pool and assigns stored correction data to channels.

See Section 4.5.3, "Calibration pool", on page 174 for background information.

Access: Channel – [Cal] > "Use Cal" > "Cal Manager"



Channel State

The "Channel State" table shows all channels in the active recall set together with their current calibration. Channels can use either the active channel calibration (if available), a previously stored user correction data or the factory system error correction (indicated as '--').

Remote command:
n/a

Add / Add All / Replace / Apply / Apply to All

The buttons between the tables are used to modify the calibration pool and apply calibration data sets (cal groups) to channels:

- "Add" copies the correction data of the selected channel to the cal pool, generating a new pool member (cal group).
- "Add All" copies the correction data of all channels to the cal pool, generating new pool members (cal groups).
- "Replace" overwrites a cal group with new correction data.
- "Apply" assigns the selected cal group to the selected channel.
- "Apply to All" assigns the selected cal group to all channels in the "Channel State" table.

For channels that are linked to a "Cal Group" (using "Apply" or "Apply to All"), a new calibration overwrites the cal group data and hence affects all channels that are also linked to this cal group. An "Overwrite Warning" is displayed in this case. To continue with the calibration, confirm by using button "Overwrite Current File?" or "Resolve Pool Link / Remove Pool Link" on page 659.

Remote command:
MMEMory:STORe:CORRection
MMEMory:LOAD:CORRection
MMEMory:LOAD:CORRection:MERGe

Pool / Delete from Pool
The "Pool" table shows all correction data sets <CalGroup_name>.cal in the directory C:\Users\Public\Documents\Rohde-Schwarz\Vna\Calibration\Data. The name of a pool data set can be modified directly in the corresponding table cell.

"Delete from Pool" deletes a cal group file from the pool. **Note** however, that calibrations being used in any of the opened recall sets cannot be deleted.

Remote command:
MMEMory:DELete:CORRection

Preset User Cal
Selects a cal group from the pool that is activated during a user-defined preset.

A "Preset User Cal" is particularly useful for scenarios involving External switch matrices, because with switch matrices user correction is indispensable.

Remote command:
SYSTem:PRESet:USER:CAL

Resolve Pool Link / Remove Pool Link
Deletes a link between the selected channel and a "Cal Group" (previously created using "Apply" or "Apply to All"). With "Resolve Pool Link", the cal group data are still used as a channel calibration ("Channel Cal") for this channel. With "Remove Pool Link", the channel calibration is removed.

Remote command:
MMEMory:LOAD:CORRection:RESolve

Ch<n> Calibration Properties/Cal Group <n> Properties
Displays the basic channel settings and the properties of the system error correction and the power correction for the channel (calibration group) selected in the "Channel State" ("Pool") table.

"Ch<n> Calibration Properties" also indicates settings mismatches (between the selected channel and the channel setup that was used during calibration) that lead to a "Cal Off". Currently this indication is limited to settings related to Parallel Measurements with frequency offset and Freq Sweep Mode.

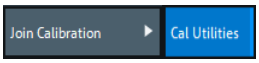
In addition, it is stated whether sweep data are available for the selected calibration.

Remote command:
[SENSe<Ch>:]CORRection:DATE?
[SENSe<Ch>:]CORRection:DATA:PARameter<Sfk>?
[SENSe<Ch>:]CORRection:DATA:PARameter<Sfk>:PORT<PhyPt>?
[SENSe<Ch>:]CORRection:DATA:PARameter:COUNt?
[SENSe<Chn>:]CORRection:SSTate?

5.11.5 Cal Utilities tab

Provides a dialog helping you to join the two calibrations.

5.11.5.1 Controls on the Cal Utilities tab



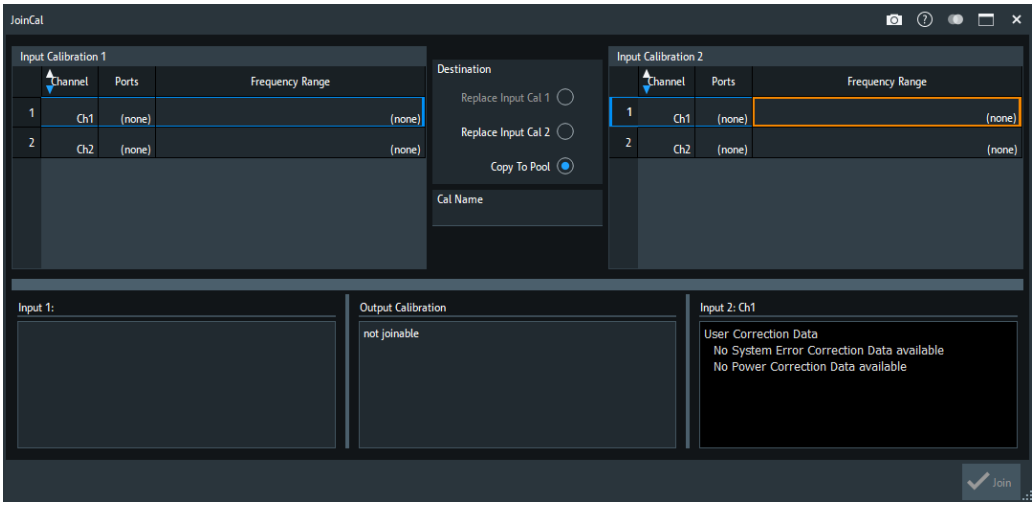
Join Calibration
Opens the JoinCal dialog.

5.11.5.2 JoinCal dialog

Allows you to join existing calibrations.

Access: Channel – [Cal] > "Cal Utilities" > "JoinCal"

Background information
See Section 4.5.10, "Joining calibrations", on page 197.



Input Calibration1/Input Calibration2/Cal Name
Select the input calibrations to be joined, and specify a name for the resulting calibration

Remote command:
`SYSTem:CORRection:JOINcal`

5.11.6 METAS Cal tab

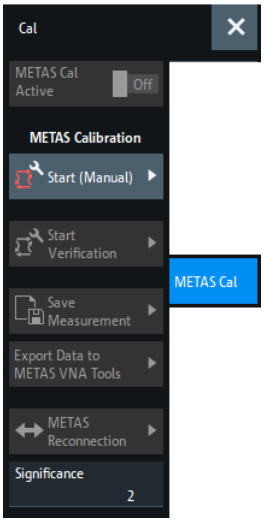
The controls on the "METAS Cal" tab allow you to set up the instrument for real-time measurement uncertainty analysis. It is only visible if option R&S ZNB-K50 "Measure-

ment Uncertainty Analysis" and METAS VNA Tools (version 2.2.3 or higher) are installed on the instrument.

For background information, see Section 4.7.7, "Measurement uncertainty analysis", on page 248.

- If external switch matrices are used, measurement uncertainty analysis is **not supported**.
- METAS calibration and verification is not fully supported via remote control. Some actions must be performed from the VNA GUI.

5.11.6.1 Controls on the METAS Cal tab



METAS Cal Active
After a successful METAS calibration, this control enables or disables the uncertainty calculation for the related channel.

- If enabled, measurement results and uncertainties are calculated by METAS VNA Tools.
- If disabled, the VNA calculates the measurement results based on standard system error correction data (without uncertainties)

Remote command:
`[SENSe<Ch>:]CORRection:METas[:STATe]`

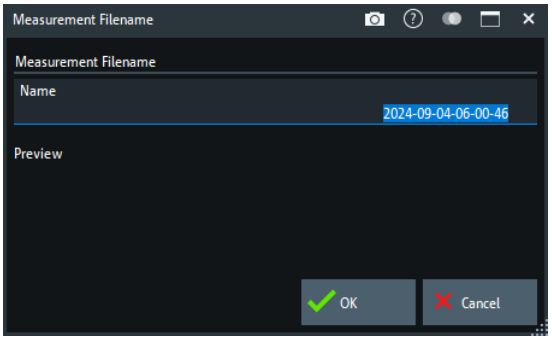
METAS Calibration – Start... (Manual)
Opens the METAS Calibration (Manual) dialog. A successful METAS calibration is activated immediately.

METAS calibration using calibration units is not supported.

Start Verification
While a METAS calibration is active, this button opens the METAS Verification dialog.

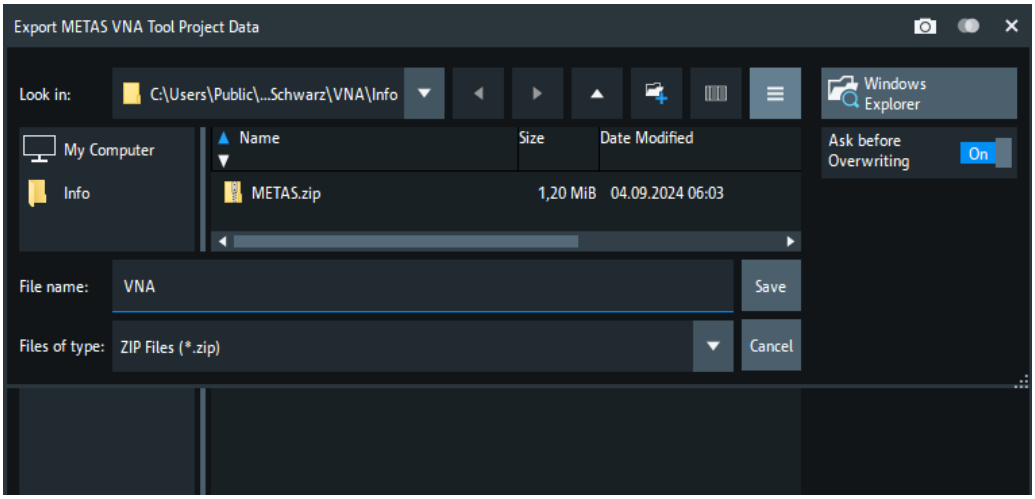
Save Measurement
Allows you to save the current measurement results to the temporary METAS VNA Tools project that is created under the hood during the METAS calibration. This project can be saved using the Export Data to METAS VNA Tools function.

Opens the "Measurement Filename" dialog that lets you specify a prefix for the name of the generated file. A timestamp of the related measurement is always appended.



If no measurement data are available, the firmware displays a message box instead of the "Measurement Filename" dialog:

Export Data to METAS VNA Tools
Saves the current state of the METAS VNA Tools project that is created with the METAS calibration.



The exported ZIP file also contains the characterizations of the standards that were used for calibration and verification. The characterization data are kept as static resources in the METAS VNA Tools database. In summary, the exported ZIP file includes everything that is required to perform an offline analysis using any METAS VNA Tools installation, not only the one on the VNA.

Note: The exported file can be directly unzipped to the METAS VNA Tools user data folder (default: `C:\Users\Public\Documents\Metas.Vna.Tools`). After (re-)opening the METAS VNA Tools, the exported project and its associated static data are available for offline analysis.

METAS Reconnection
Opens the METAS Reconnection dialog.

This button is only enabled, if a METAS cal is active.

Significance
Defines the width of the uncertainty band, expressed as multiples of the standard deviation.

Default is 2 (95% confidence interval).

Remote command:
`[SENSe<Ch>:]CORRection:METas:UNCertainty`

5.11.6.2 METAS Calibration (Manual) dialog

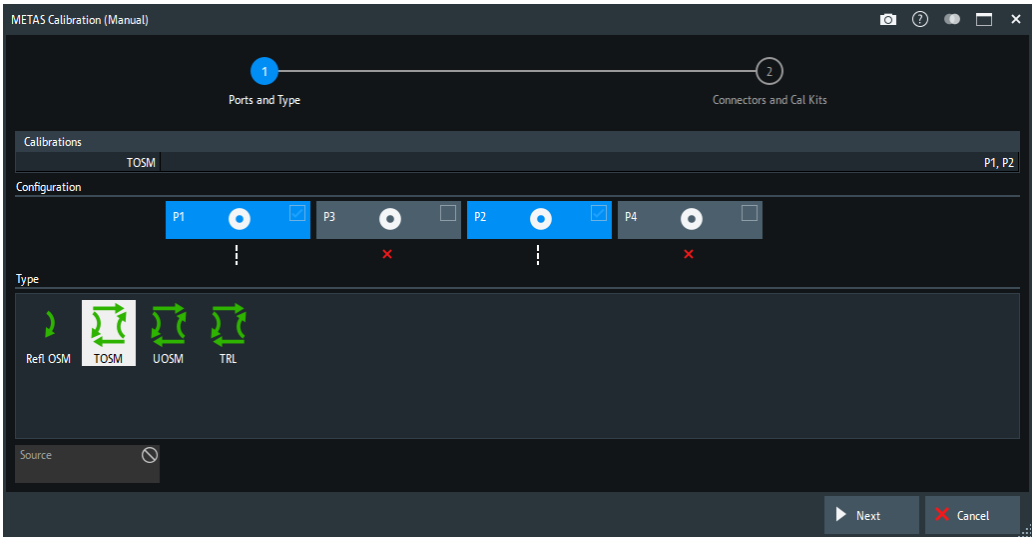
This dialog allows you to set up and perform a METAS Calibration of the active channel.

Access: Channel – [Cal] > "METAS Cal" > METAS Calibration – "Start (Manual)"

Only a single VNA channel can be METAS calibrated at a time. If you have a METAS calibration for another channel (no matter in which recall set), this METAS calibration is deleted when you apply the new one.

Step 1: Ports and Type

In step 1, choose the ports to be calibrated and the calibration type to be performed.



Only calibration types supported by METAS VNA Tools are available.

Ports
Selects the test ports to be calibrated.

Note: Calibration and port de-/activation.

The analyzer firmware automatically activates/deactivates ports during/after a (successful) calibration:

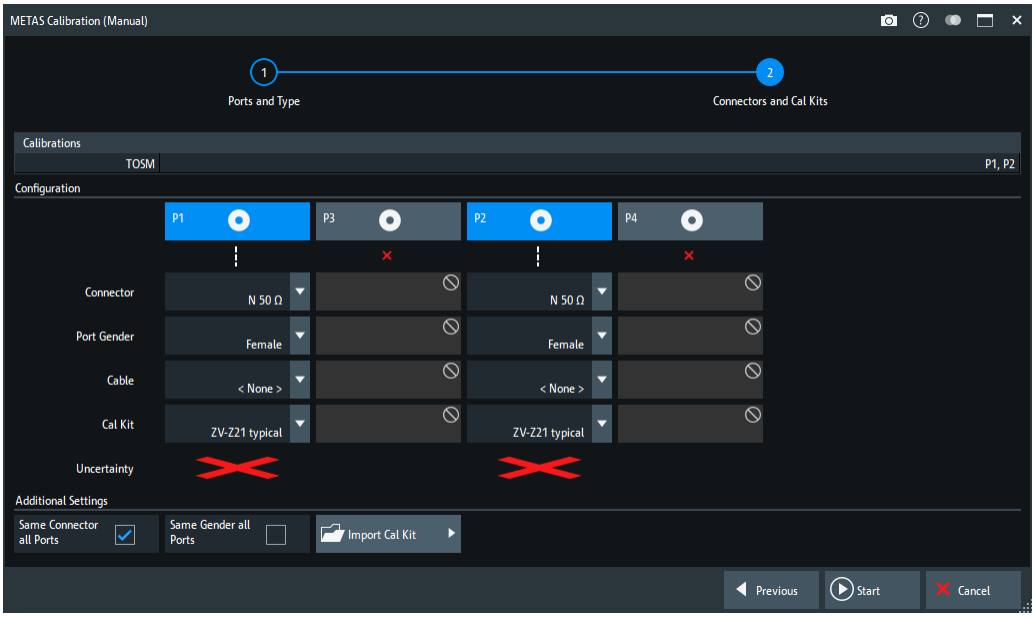
- Calibrated ports that were previously disabled, are automatically enabled as single-ended logical ports.
- An uncalibrated port that is not used by a measurement (i.e. the port is not required by any trace of the related channel) is disabled.

Remote command:
The port parameters in many calibration commands define the calibrated ports.

Type
Selects the calibration type.

Remote command:
`[SENSe<Ch>:]CORRection:COLLect:METHod:DEFine <CalName>, MTOSm | MUOSm | MOSM | MTRL, ...`

Step 2: Connectors and Cal Kits



Connector/Port Gender
Specifies the connector and gender of the ports selected in Step 1: Ports and Type.

Note that METAS compatible file formats do not contain information about the related connector type, so you have to make sure to choose a suitable Cal Kit.

Remote command:
See Connector / Port Gender

Cable
Allows you to specify which cables are connected to the related ports.

The cable models are read from the METAS VNA Tools database. Additional cable models can be added on a project-specific basis.

Remote command:
`[SENSe:]CORRection:METas:CABLe:CATalog?`
`[SENSe<Ch>:]CORRection:METas:CABLe:TYPE`

Cal Kit/Uncertainty
Allows you to specify the cal kits to be used for the selected ports. For meaningful uncertainty calculations, use a cal kit that is characterized with uncertainties.

Regular cal kits (without uncertainties) are also offered for selection. For these cal kits, zero uncertainty is assumed, which results in an unrealistic METAS calibration quality.

The "Uncertainty" control indicates whether the selected characterization actually comprises uncertainties.

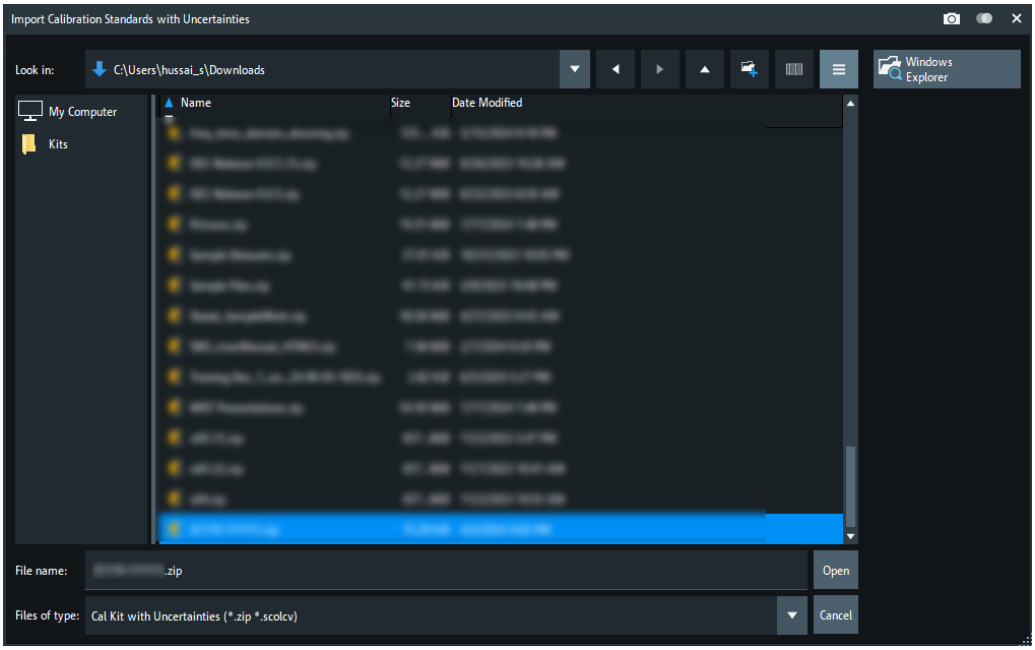
Remote command:
`[SENSe:]CORRection:CKIT:SELect`

Same Connector all Ports/Same Gender all Ports
See "Same Connector All Ports / Same Gender All Ports" on page 587.

Import Cal Kit...
Opens the "Import Calibration Standards with Uncertainties" dialog that allows you to import cal kits with (or without) uncertainties.

A "CalKit with Uncertainties" can be one of the following:
- A `zip` file containing calibration standard characterizations in a format supported by METAS VNA Tools (`sdatb`, `sdatx`, `sdatcv`, `calstd`)
- A `scolcv` file

See the document "Data Formats" on the METAS VNA Tools internet page (https://www.metas.ch/metas/en/home/fabe/hochfrequenz/vna-tools.html, section "VNA Tools Documentation".



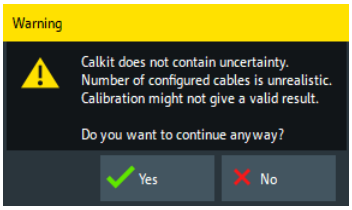
The imported cal kit is assigned to the selected Connector type; the name of the ZIP file is used as the cal kit name.

The name of a characterization file inside the zip container must indicate:

- The standard's type (Open, Short, Load, Match, Line, Reflect, Through, Sliding Load, Sliding Match)
- The standard's gender (f, m, ff, fm, mf, mm), if applicable.

Imported cal kits with uncertainties are preserved and added to the Cal Kit selection combo-box. There and in the Calibration Kits dialog, they appear with an "Uncertainty" flag.

Start
Proceeds to Step 3: Calibration. If the configuration is not plausible, a warning is displayed:



Step 3: Calibration

In step 3, the calibration is performed. The procedure is the same as for regular calibrations of the selected type (see "Step 3: Calibration" on page 587).

On "Apply", the raw calibration data are transferred to METAS VNA Tools, which take over the calculation of measurement results and uncertainties: METAS Cal Active is enabled automatically.

In parallel to the "METAS Calibration", a regular VNA calibration without uncertainties is created. This calibration can be activated if "METAS Cal Active" is disabled.

5.11.6.3 METAS Verification dialog

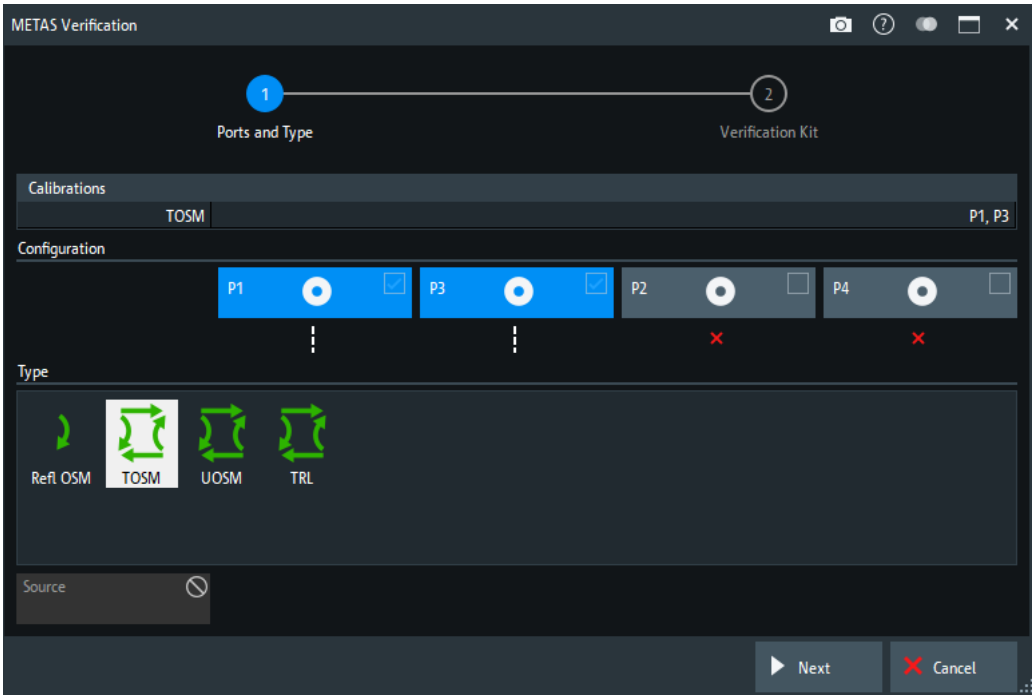
This dialog allows you to set up and perform a Verification of a METAS Calibration.

For background information, see "Verification" on page 250.

Access: Channel – [Cal] > "METAS Cal" > "Start Verification"

Step 1: Verification Kit

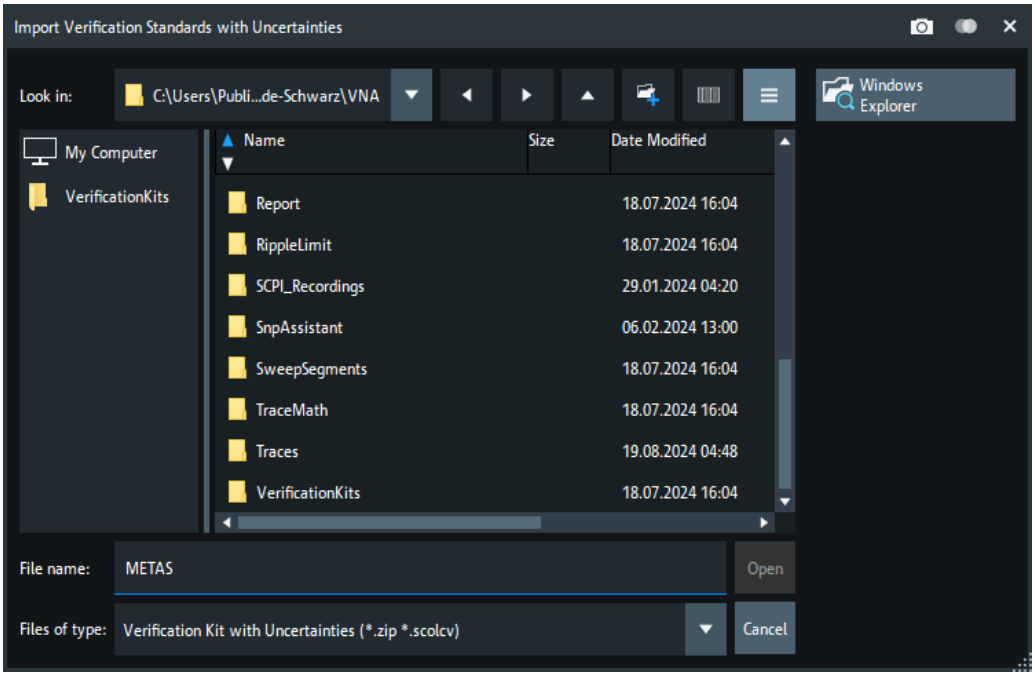
Allows you to select the verification kit to be used. All other settings are taken from the active METAS calibration and cannot be changed.



Verification Kit
Allows you to specify the verification kits to be used for the related ports.

Verification standards are characterized individually, including uncertainties. So initially no verification kits are defined. Use Import Verification Kit... to import your verification kit.

Import Verification Kit...
Opens the "Import Verification Standards with Uncertainties" dialog.



A "Verification Kit with Uncertainties" is a ZIP file containing verification standard characterizations in a format supported by METAS VNA Tools (`sdatb`, `sdatx`, `sdatcv`). See the document "Data Formats" on the METAS VNA Tools internet page (https://www.metas.ch/metas/en/home/fabe/hochfrequenz/vna-tools.html, section "VNA Tools" > "Installer and Documentation").

The imported verification kit is assigned to the selected Connector type; the name of the ZIP file is used as the verification kit name.

The name of a characterization file inside the zip container must indicate:

- The standard's type (offset short, mismatch, 20 dB Att, 40/50 dB Att, airline, mismatch airline, att, attenuator, stepped thru)
- The standard's gender (f, m, fm, ff, mf, mm), if applicable

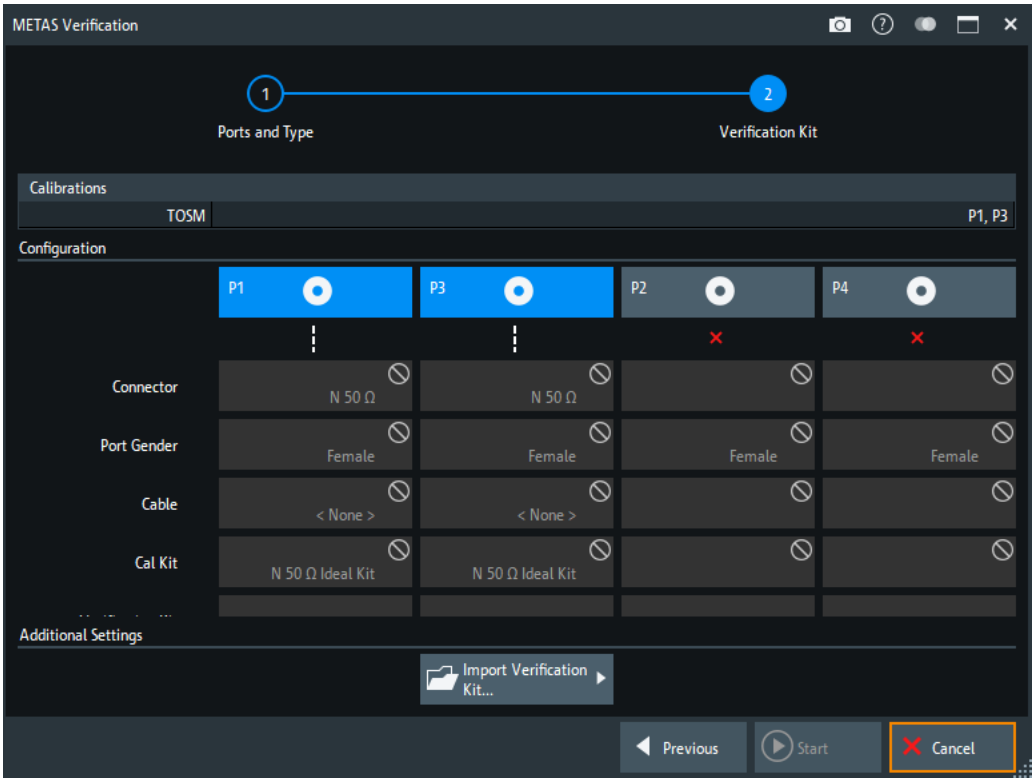
Imported verification kits with uncertainties are preserved and added to the Verification Kit selection combo-box.

Step 2: METAS Verification

Step 2 allows you to perform the verification.

By selecting a button representing a verification standard, the required measurements are performed. The raw measurement data are handed over to METAS VNA Tools that returns the calibrated results and uncertainties and the pass/fail states of the respective verifications. The latter are indicated on the trace fields to the right of the verification standard buttons (red = failed).

The verification standard buttons indicate the verification states of the respective verification standards ("V" = not verified yet, "✓" = passed, "X" = failed). Use the Collapse button to toggle a condensed view focusing on these verification states.



Collapse
In collapsed state, the "METAS Verification" dock widget presents a condensed view, focusing on the verification results for the available verification standards.

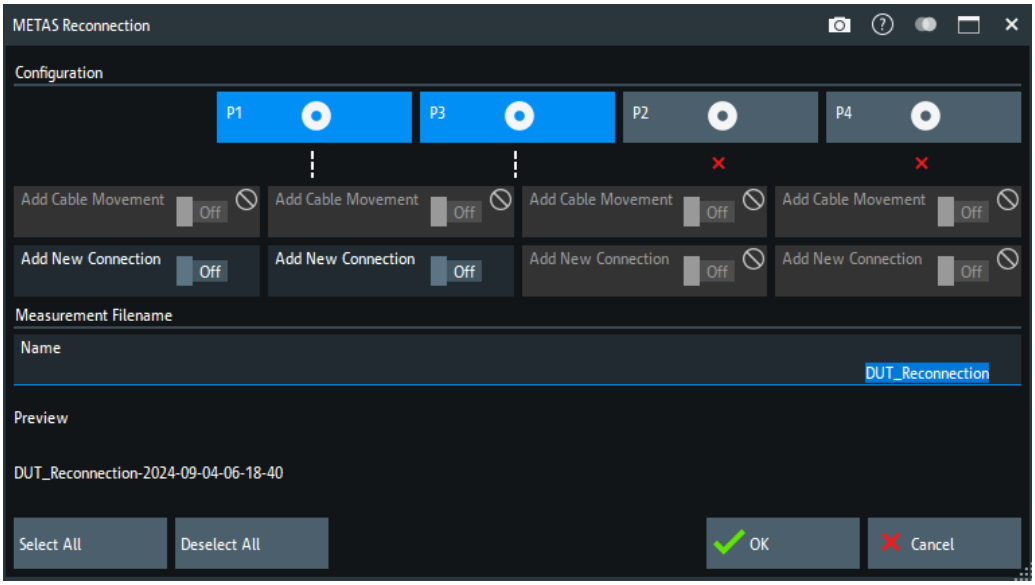
Save Verification Data
Same as Export Data to METAS VNA Tools. The exported ZIP file now also contains the METAS verification configuration, and the verification measurement data obtained so far.

5.11.6.4 METAS Reconnection dialog

While a METAS calibration is active, you can notify the METAS VNA Tools about additional reconnections and/or cable movements. The latter is only applicable if cables were configured during the setup of the METAS calibration.

Access: main toolbar icon or

Channel – [Cal] > "METAS Cal" > "METAS Reconnection"



Both reconnections and cable movements contribute to the overall measurement uncertainty. In the METAS VNA Tools project created during the METAS calibration), these events are registered in the so-called measurement journal. In addition, the results measured immediately after the event report are persisted in the project.

The effect of a reconnection depends on the selected connector, the effect of a cable movement depends on the selected cable (on their respective models, to be precise).

For the METAS calibration, the VNA firmware reports the minimum possible number of reconnections (depending on the selected calibration type) and no cable movements. All other events of these types must be reported manually, using the "METAS Reconnection" dialog.

Add New Connection/Add Cable Movement
While a METAS calibration is active, these buttons notify the METAS VNA Tools about additional connector reconnections and/or cable movements. The latter is only applicable if cables were configured during the setup of the METAS calibration.

Remote command:
`[SENSe<Ch>:]CORRection:METas:CABLe:REConnection`
`[SENSe<Ch>:]CORRection:METas:CABLe:MOVement`

Measurement Filename
Allows you to configure the name of the measurement data file generated with the measurement journal entry. For uniqueness, a timestamp is always appended.

5.12 Channel Config softtool

The "Channel Config" functions select, create and delete channels, configure the source and receive ports of the R&S ZNB, and optimize the measurement process.

Access: Channel – [Channel Config] hardkey

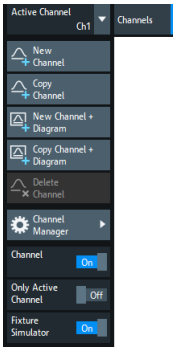
5.12.1 Channels tab

Allows you to create and delete channels, to modify the channel state, and to select a channel as the active channel.

You can monitor the channel activity using the `OUTPut<Ch>:UPORt[:VALue]` command and the output signals at pins 8 to 11 of the USER PORT connector.

Background information
Refer to Section 4.1.3.3, "Active and inactive traces and channels", on page 82.

5.12.1.1 Controls on the Channels tab



The buttons in the "Channels" tab open the following dialogs:

- "Channel Manager": see Section 5.12.1.2, "Channel Manager dialog", on page 673
- "RFFE": see Section 5.12.1.3, "RFFE Config dialog for R&S ZN-B15/-Z15 Var. 03", on page 674

Active Channel
Selects an arbitrary channel of the active recall set as the active channel. This function is disabled if the current recall set contains only one channel.

If one or several traces are assigned to the selected channel, one of these traces becomes the active trace.

The order of all channels in a recall set is given by the channels' creation time. By default, the channels are named Ch1, Ch2, ... so that Ch<n – 1> precedes Ch<n>. This order is always maintained, even if channels are renamed, invisible (because no traces are assigned to them) or distributed over several diagram areas.

Tip: You can also select a line in the channel list to activate the corresponding channel.

Remote command:
The numeric suffix <Ch> appended to the first-level mnemonic of a command selects a channel as the active channel.

New Channel
Creates a channel and a trace with default settings, and assigns the new trace to the active diagram area.

The new channel is named Ch<n>, where <n> is the largest of all existing channel numbers plus one. The name can be changed in the "Channel Manager" dialog.

Tips:
- To create a channel and a trace with default settings and to display the trace in a *new* diagram area, use New Channel + Diagram.
- Use Add Trace to create a trace in the *active* channel.

Remote command:
```
CALCulate:PARameter:NEW:CHANnel [OFF]
```

Copy Channel
Copies the active channel with all its settings (including a possible channel calibration) and traces. The new traces are displayed in the active diagram area.

The new channel is named Ch<n>, where <n> is the largest of all existing channel numbers plus one. The name can be changed in the "Channel Manager" dialog.

Tips: Use Copy Channel + Diagram to display the copied traces in a *new* diagram area.

Remote command:
```
CALCulate<Ch>:PARameter:COPY:CHANnel [OFF]
```

New Channel + Diagram
Creates a channel and a trace with default settings, and assigns the created trace to a new diagram area.

The new channel is named Ch<n>, where <n> is the largest of all existing channel numbers plus one. The name can be changed in the "Channel Manager" dialog.

Tip: To create a channel and a trace with default settings and to display the trace in the *active* diagram area, use New Channel.

Remote command:
`CALCulate:PARameter:NEW:CHANnel ON`

Copy Channel + Diagram
Copies the active channel with all its settings (including a possible channel calibration) and traces, and displays the created traces in a new diagram area.

The new channel is named Ch<n>, where <n> is the largest of all existing channel numbers plus one. The name can be changed in the "Channel Manager" dialog.

Tips: Use Copy Channel to display the new traces in an *existing* diagram area.

Remote command:
`CALCulate<Ch>:PARameter:COPY:CHANnel ON`

Delete Channel
Deletes the current channel including all the traces assigned to it and removes all display elements related to the channel from the diagram area. "Delete Channel" is disabled if the recall set contains only one channel: In manual control, each recall set must contain at least one diagram area with one channel and one trace.

Tips: Use the "Undo" function to restore a channel that was unintentionally deleted.

Remote command:
`CONFigure:CHANnel<Ch>[:STATe] OFF`

Channel On
Toggles the measurement state of the active channel.

Remote command:
`CONFigure:CHANnel<Ch>:MEASure[:STATe]`

Only Active Channel On
If enabled, only the active channel is measured; all other channels are switched off temporarily and dynamically.

Temporarily means: If "Only Active Channel On" is disabled, then all channels return to their original Channel On state.

Dynamically means: If another channel becomes the active one, the firmware adjusts the temporary ON/OFF state accordingly.

Remote command:
`CONFigure:CHANnel:MEASure:ACTive[:STATe]`

Fixture Simulator
The "Fixture Simulator" switch deactivates or activates the configured deembedding, embedding, balanced ports, and port impedance settings for the selected channel.

When "Fixture Simulator" is **deactivated:**
- All balanced ports are resolved to single ended ports
- All port impedances are set to default

- All de/embeddings are disabled

At the GUI, the "Balanced Ports" dialog and the de/embedding tabs and dock widgets are disabled. Related remote commands generate an error.

When "Fixture Simulator" is **reactivated**, the situation before the deactivation is restored.

Note: The "Offset" and "One Way Loss" settings are **not** affected by the "Fixture Simulator" switch. Use the "All Compensation Off"/"All Compensation On" functions of the "Fixture Compensation" dialog to de/activate these compensations as well (see Section 5.13.2.2, "Fixture Compensation dialog", on page 741).

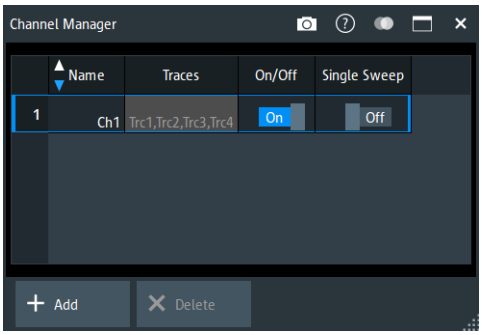
Remote command:
```
CALCulate<Ch>:TRANsform:VNETworks:FSIMulator[:STATe]
```

5.12.1.2 Channel Manager dialog

The "Channel Manager" dialog allows you to rename channels and to change their measurement state and sweep mode.

Access: Channel – [Channel Config] > "Channels" > "Channel Manager"

Background information
Refer to Section 4.1.3.3, "Active and inactive traces and channels", on page 82.



Channel table
The rows and columns of the channel table represent the existing channels (rows) together with certain editable (white) or non-editable (gray) properties (columns).

- "Name" indicates the name of the related channel.
- "Traces" indicates the names of all traces assigned to the related channel.
- "On/Off" toggles the measurement state of the related channel.
- "Single Sweep" toggles between "Continuous" and "Single" sweep mode (see "Continuous / Single" on page 559).

Remote command:
```
CONFigure:CHANnel:CATalog?
CONFigure:CHANnel<Ch>:NAME
CONFigure:CHANnel<Ch>:NAME:ID?
CONFigure:CHANnel<Ch>:MEASure[:STATe]
INITiate<Ch>:CONTinuous
```

Add / Delete
The buttons below the channel table add and delete channels.

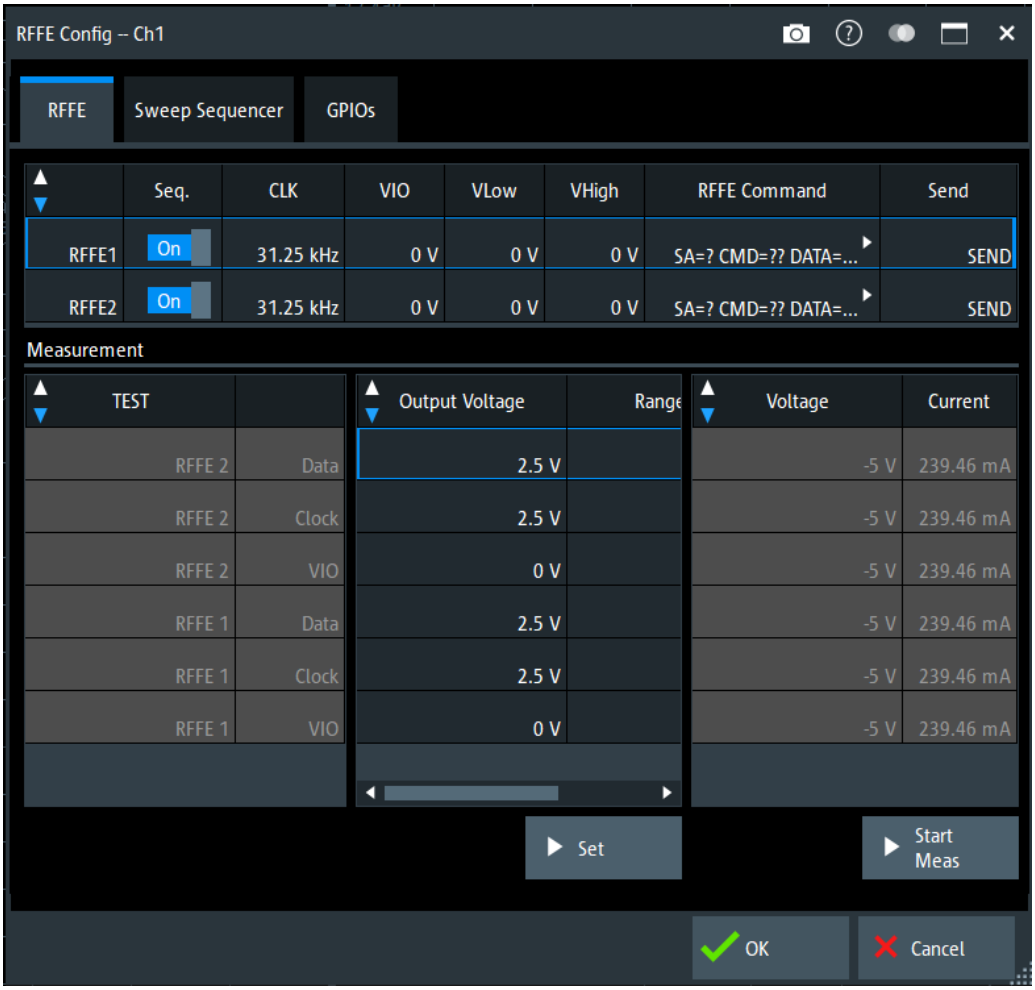
- "Add" adds a new channel to the list. The new channel is named Ch<n>, where <n> is the largest of all existing channel numbers plus one.
- "Delete" deletes the channel selected in the table. This button is disabled if the setup contains only one channel: In manual control, each setup must contain at least one diagram area with one channel and one trace.

Remote command:
`CONFigure:CHANnel<Ch>[:STATe]`

5.12.1.3 RFFE Config dialog for R&S ZN-B15/-Z15 Var. 03

Controls the channel-specific setup of the two RFFE bus interfaces and GPIO ports provided by **variant 03** of the internal/external RFFE GPIO extension board R&S ZN-B15/-Z15.

Access: Channel – [Channel Config] > "Channels" > "RFFE"



If the instrument is equipped with **variant 02** of the extension board, a slightly different user interface is shown. The "Measurement" columns are hidden and the remaining content of the RFFE and GPIO tab are presented on a single "Control" tab.

Background information

Refer to Section 4.7.20, "RFFE GPIO interface", on page 260.

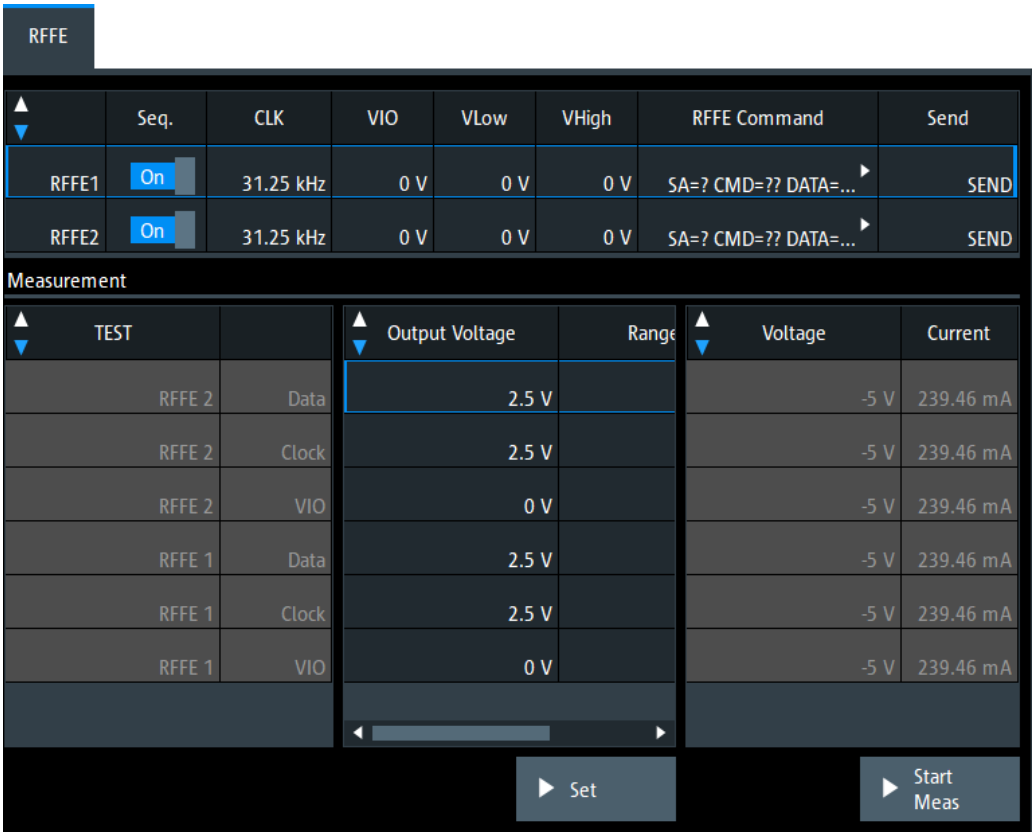
For more details about the voltage range, clock frequency ranges and their steps sizes, refer to Section 13.3.5, "RFFE GPIO interface", on page 1715.

RFFE tab

The "RFFE" tab is divided into two parts:

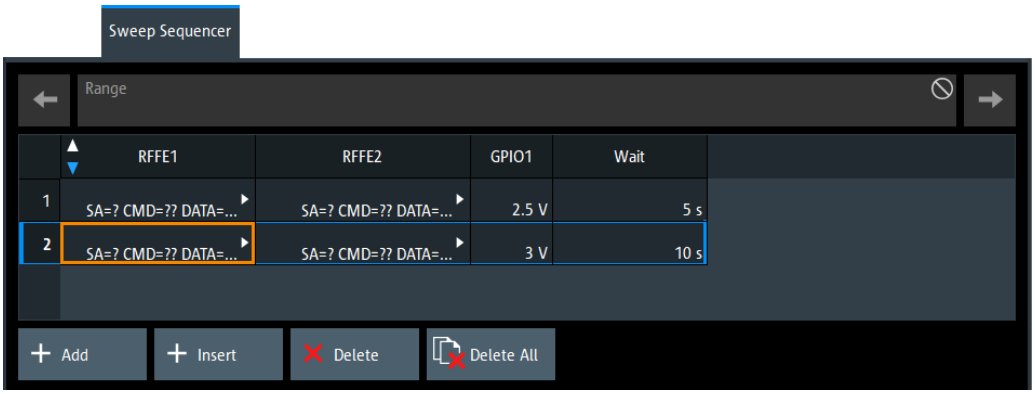
- The upper part gives access to the RFFE interface settings and allows manual command execution and result display (see "Basic RFFE interface settings and command execution" on page 677)
- The lower part allows you to define and apply test voltages and to execute voltage and current measurements on the RFFE pins (see "GPIO voltage and current measurements" on page 681)

The "Set" button activates the "Output Voltage" and "Range" ("Shunt" resistance) settings. The "Meas" button starts the voltage and current measurements.



Sweep Sequencer tab

Gives access to the "Sweep Sequencer" functionality, see Section 5.12.1.6, "Sweep Sequencer", on page 683.

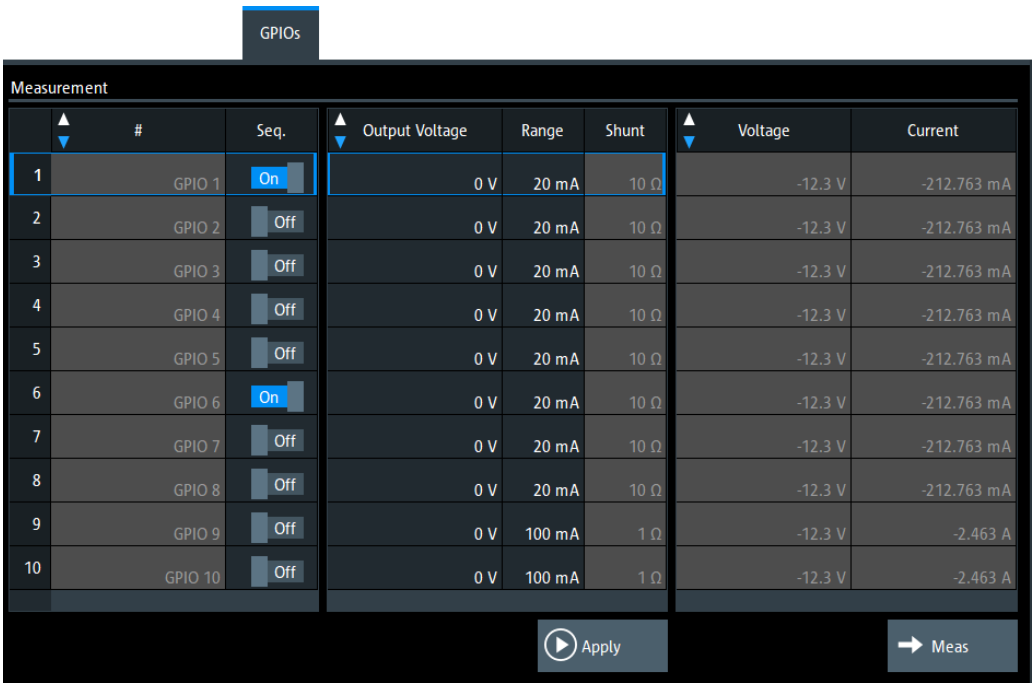


GPIOs tab

The "GPIOs" tab is split into two parts:

- The left part of the table area (up to column "Output Voltage") allows you to define and apply the GPIO pin voltages (see "Basic GPIO configuration" on page 680).
- The right part allows you to define and execute voltage and current measurements on the GPIO pins (see "GPIO voltage and current measurements" on page 681)

The "Apply" button activates both the "Output Voltage" and "Range" ("Shunt" resistance) settings. The "Meas" button starts the voltage and current measurements.



5.12.1.4 RFFE interface configuration

Access: RFFE tab of the "RFFE Config" dialog

Basic RFFE interface settings and command execution

The RFFE config table gives access to the channel-specific setup of the two RFFE bus interfaces RFFE1 and RFFE2.

If at least one of the RFFE commands reads back values, an additional result area is shown below the table. For each of these read-back commands, it displays the respective read count or, after the command was sent, the last received response (if successfully read).



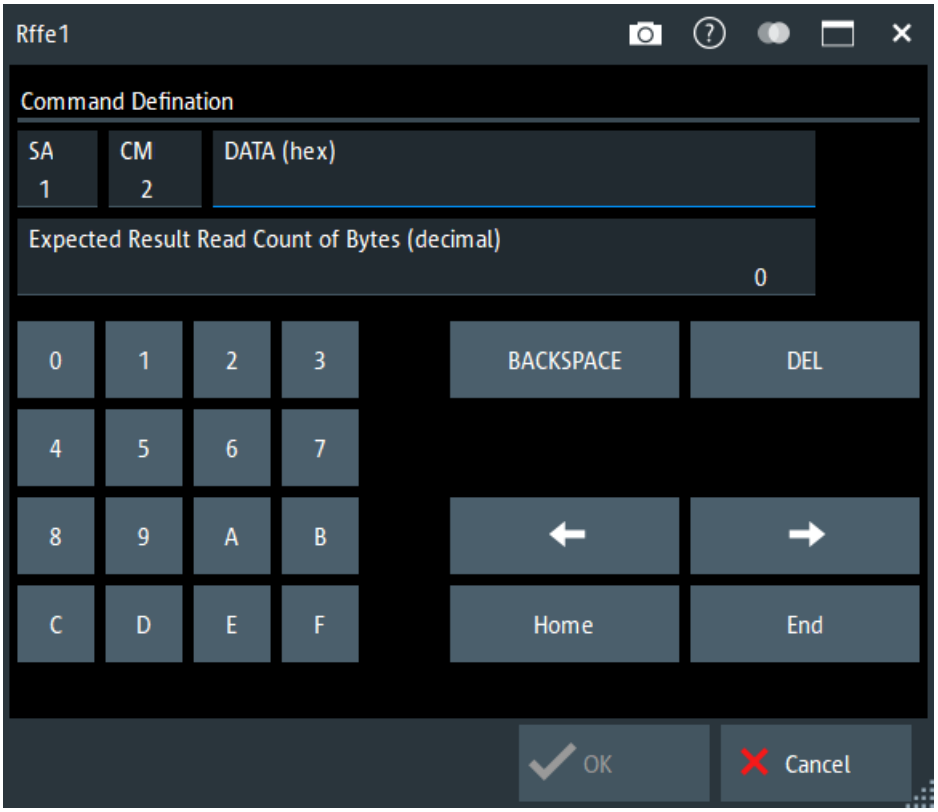
Seq.
Enables/disables the corresponding RFFE interface in the Sweep Sequencer (see Section 5.12.1.6, "Sweep Sequencer", on page 683).

Remote command:
`CONTrol<Ch>:RFFE<Bus>:SETTings[:STATe]`

CLK, VIO, VLow, VHigh
These columns give access to the physical properties of the RFFE interfaces: clock rate ("CLK"), supply voltage ("VIO") and the voltage levels of the data signal SDATA ("VLow") and clock signal SCLK ("VHigh").

Remote command:
`CONTrol<Ch>:RFFE<Bus>:SETTings:FREQuency`
`CONTrol<Ch>:RFFE<Bus>:SETTings:VOLTage:IO`
`CONTrol<Ch>:RFFE<Bus>:SETTings:VOLTage:LOW`
`CONTrol<Ch>:RFFE<Bus>:SETTings:VOLTage:HIGH`

RFFE Command
The ellipsis buttons in these columns open a dialog that allows to define the RFFE command to be sent.



In this dialog, enter the following parameters (according to the device to be controlled):

- "SA": a slave address between 0 and 15 as 1 hex digit
- "CMD": a command number between 0 and 255 as 2 hex digits
- "DATA": the data part, 0 bytes to 17 bytes (hex digit **pairs**)
- "Expected Result Read Count of Bytes (decimal)": The number of bytes to be read back after the command was sent (max. 16). If set to 0, the command is a pure write command.
  In Sweep Sequencer mode, this section is hidden.

For details and background information, see the "MIPI Alliance Specification for RF Front-End Control Interface".

Remote command:
`CONTrol<Ch>:RFFE<Bus>:COMMand:DATA`

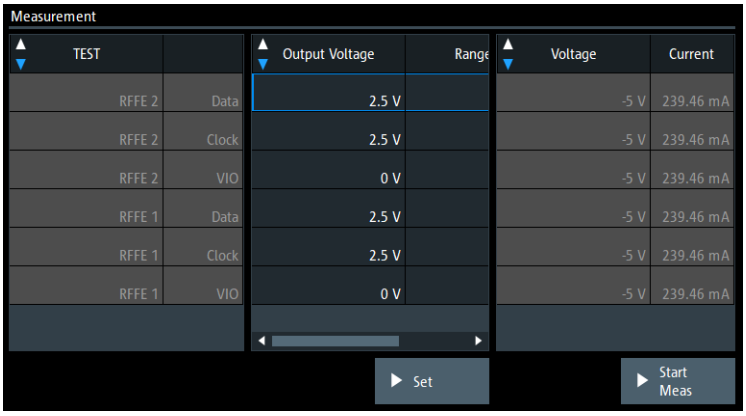
SEND
Use the "SEND" button to send the previously defined command, e.g. before starting the sweep for the related channel.

On R&S ZN-B15/Z15 **var. 03**, before the command is executed the related shunt resistance is set to its minimum possible value.

Remote command:
`CONTrol<Ch>:RFFE<Bus>:COMMand:SEND` (write-only) or
`CONTrol<Ch>:RFFE<Bus>:COMMand:SEND? <BytesToRead>` (with read-back)

RFFE interface voltage and current measurements

Defines the voltage and current measurements on the RFFE pins.



- The measurement parameters are channel-specific. However only one configuration can be measured at a time.
- Voltage and current measurements on the RFFE and GPIO pins are only possible with **Var. 03** of the extension board R&S ZN-B15/Z15 (part number 1323.9355.03 or 1325.5905.03).

Output Voltage
Defines the output voltages for the voltage/current measurements on the RFFE pins.

Note: The Data and Clock pins always use the same output voltages; their values cannot be set independently.

The "Output Voltage" and Range / Shunt settings do not take effect until Set is pressed.

Remote command:

```
CONTrol<Ch>:RFFE<Bus>:TEST:DATA
CONTrol<Ch>:RFFE<Bus>:TEST:CLOCk
CONTrol<Ch>:RFFE<Bus>:TEST:VIO
```

Range / Shunt
Defines the current range for the voltage and current measurement on the respective RFFE pin. The analyzer firmware automatically selects a suitable shunt resistance.

The Output Voltage and "Range" / "Shunt" settings do not take effect until Set is pressed.

Remote command:

```
CONTrol<Ch>:RFFE<Bus>:TEST:CLOCk:RANGe
CONTrol<Ch>:RFFE<Bus>:TEST:CLOCk:SHUNt?
CONTrol<Ch>:RFFE<Bus>:TEST:DATA:RANGe
CONTrol<Ch>:RFFE<Bus>:TEST:DATA:SHUNt?
CONTrol<Ch>:RFFE<Bus>:TEST:VIO:RANGe
CONTrol<Ch>:RFFE<Bus>:TEST:VIO:SHUNt?
```

Set
Use the "Set" button to apply all RFFE configuration changes and to activate the configured voltage and Range / Shunt settings on the RFFE pins.

Remote command:
```
CONTrol<Ch>:RFFE:TEST:OUTPut
```

Start Meas
Starts the voltage and current measurement.

Note that "Start Meas" does not automatically activate the "Output Voltage" and "Range" / "Shunt" settings of the current channel. Use Set to activate them manually.

The measurement (=sampling) time can only be set via remote command. It is not channel-specific and applies to RFFE and GPIO measurements.

Remote command:
```
CONTrol<Ch>:RFFE:TEST:SENSe:TRIGger
CONTrol<Ch>:RFFE:TEST:TIME
```

Voltage, Current
Results of the voltage/current measurements on the related RFFE pins.

Remote command:
```
CONTrol:RFFE<Bus>:TEST:DATA:VOLTage?
CONTrol:RFFE<Bus>:TEST:CLOCk:VOLTage?
CONTrol:RFFE<Bus>:TEST:VIO:VOLTage?
CONTrol:RFFE<Bus>:TEST:DATA:CURRent?
CONTrol:RFFE<Bus>:TEST:CLOCk:CURRent?
CONTrol:RFFE<Bus>:TEST:VIO:CURRent?
```

5.12.1.5 GPIO configuration

Access: GPIOs tab of the "RFFE Config" dialog

Basic GPIO configuration

The (left part of the) GPIO config table gives access to the channel-specific GPIO voltage settings.

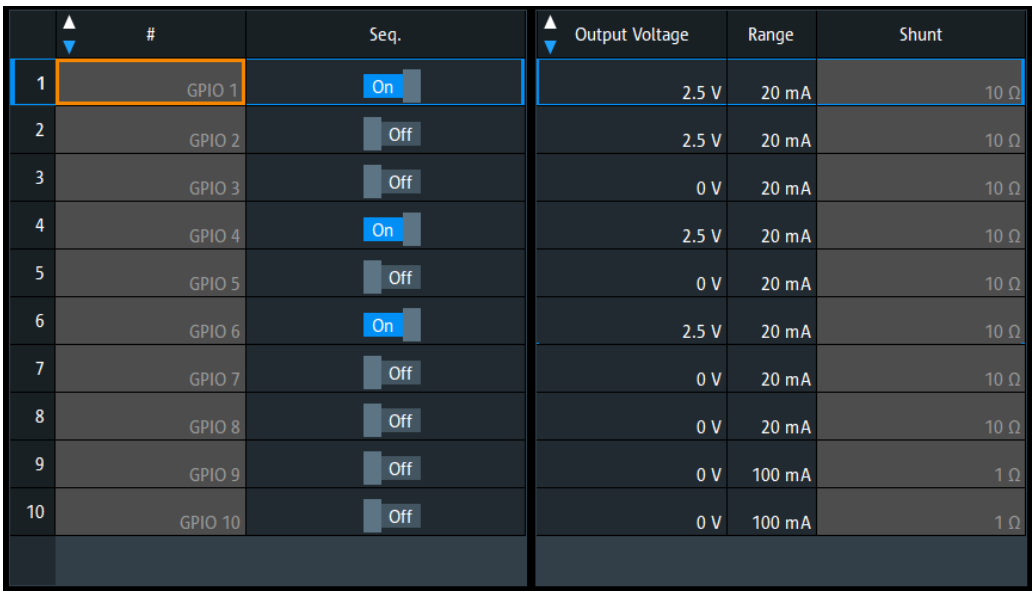


Figure 5-69: Basic GPIO Configuration for HW Var. 03

Seq.
Enables/disables the corresponding GPIO pin in the Sweep Sequencer (see Section 5.12.1.6, "Sweep Sequencer", on page 683).

Remote command:
`CONTrol<Ch>:GPIO<Port>[:STATe]`

Voltage / Output Voltage
Sets the (default) voltage of the respective GPIO pin for R&S ZN-B15/-Z15 Var. 02 / Var. 03.

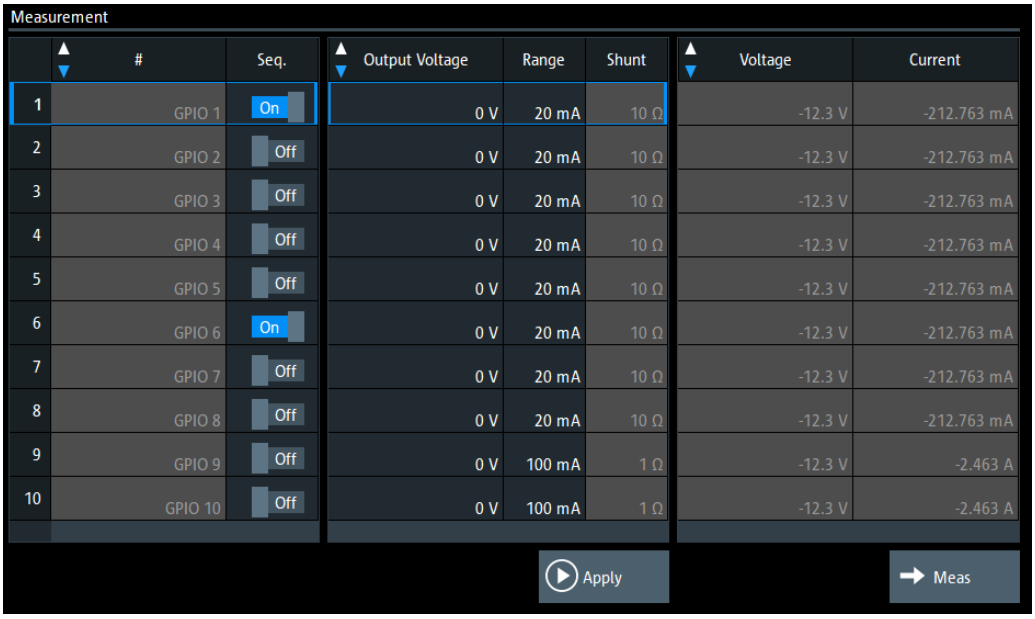
Remote command:
`CONTrol<Ch>:GPIO<Port>:VOLTage[:DEFault]`

Apply
Use the "Apply" button to activate the configured voltage (and Range / Shunt) settings to the GPIO pins.

Remote command:
`CONTrol<Ch>:GPIO:VOLTage:OUTPut`

GPIO voltage and current measurements

Defines and executes the voltage and current measurements on the GPIO pins.



- The measurement parameters are channel-specific. However only one configuration can be measured at a time.
- Voltage and current measurements on the RFFE and GPIO pins are only possible with **Var. 03** of the extension board R&S ZN-B15/-Z15 (part number 1323.9355.03 or 1325.5905.03).
  The high-resistance configuration of GPIO pins 9 and 10 requires FPGA version 6.1.0 or higher. For older versions of the R&S ZN-B15/Z15 Var. 03, pins 9 and 10 have the same current range as pins 1 to 8 (see "Range / Shunt" on page 682).
  To check for an equipped RFFE/GPIO interface's part number and "Product Index" version, see the Hardware tab of the "Info" dialog.

Range / Shunt
"Range" defines an upper bound of the current to be measured on the respective GPIO pin. The analyzer firmware automatically selects a suitable shunt resistance.

Possible ranges for ports 1 to 8 are { $2 \cdot 10^n$ µA | n=1,...,5 }.

For R&S ZN-B15/Z15 Var. 03 with FPGA version 6.1.0 or higher, Pins 9 and 10 can be configured as follows:
- 0 A "high resistance" range (shunt resistance 100 MΩ)
- 100 mA range (shunt resistance 1 Ω)

For older versions, pins 9 and 10 have the same current range as pins 1 to 8.

The Voltage / Output Voltage and "Range" / "Shunt" settings do not take effect until Apply is pressed.

Remote command:
`CONTrol<Ch>:GPIO<Port>:RANGe`
`CONTrol<Ch>:GPIO<Port>:SHUNt?`

Meas
Starts the voltage and current measurement.

Note that "Start Meas" does not automatically activate the "Voltage" and "Range" / "Shunt" settings of the current channel. Use Apply to activate them manually.

The measurement (=sampling) time can only be set via remote command. It is not channel-specific and applies to RFFE and GPIO measurements.

Remote command:

```
CONTrol<Ch>:GPIO:SENSe:TRIGger
CONTrol<Ch>:GPIO:TIME
```

Voltage, Current

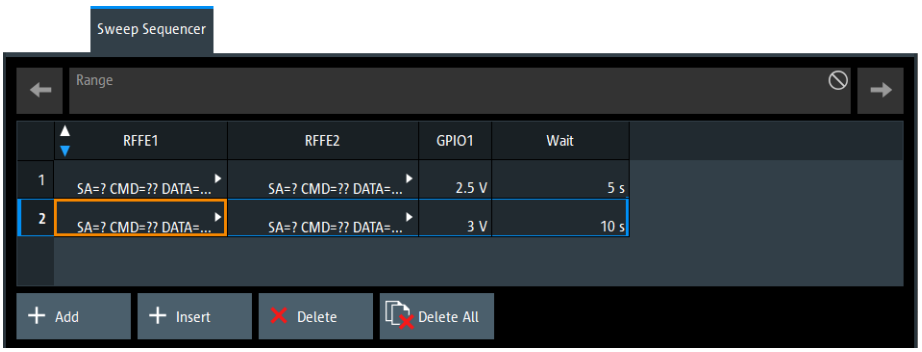
Results of the voltage/current measurements on the related GPIO pins.

Remote command:

```
CONTrol:GPIO<Port>:SENSe:VOLTage?
CONTrol:GPIO<Port>:SENSe:CURRent?
CONTrol:GPIO:SENSe:SUMCurrent?
```

5.12.1.6 Sweep Sequencer

Defines the RFFE commands to be executed and the GPIO voltages to be applied before each sweep (segment).



- Make sure that the required RFFE bus interfaces and GPIO ports are enabled by setting the respective "Seq." flags in the "RFFE Config" dialog (see "Basic RFFE interface settings and command execution" on page 677 and Basic GPIO configuration).
- The sequencer settings are only valid for the active channel. In a multiple channel measurement, you have to configure each channel separately.
- On R&S ZN-B15/Z15 **var. 03**, the sweep sequencer overwrites the currently applied voltages and sets the related shunt resistances to the minimum possible value (see "RFFE interface voltage and current measurements" on page 679 and "GPIO voltage and current measurements" on page 681).

Proceed as follows:

1. For segmented sweeps, select the related sweep segment (see "Range" on page 684).

2. Use the "Add" or "Insert" button to add a new step to the command/switch sequence - either at the end of the existing sequence or above the selected step, respectively.
3. Make the appropriate settings in the sweep sequencer table:
   a) Define the RFFE commands ("RFFE" columns).
   b) Set the GPIO voltages ("GPIO" columns).
   c) Specify the delays between subsequent commands/switches and before the sweep or segment start ("Wait" column).
4. Use "Delete" to remove the selected item or "Delete All" to clear or the command/switch sequence.
5. Select "OK" to apply the settings. To discard the changes, select "Cancel".

Range
For segmented sweeps, a command/switch sequence can be defined for each sweep segment. For unsegmented sweeps, only a single "Range" is available.

Remote command:
Unsegmented sweeps
`CONTrol<Ch>:SEQuence:COUNt?`
`CONTrol<Ch>:SEQuence:CLEar:ALL`
Segmented sweeps:
`CONTrol<Ch>:SEGMent<SegNr>:SEQuence:COUNt?`
`CONTrol<Ch>:SEGMent<SegNr>:SEQuence:CLEar:ALL`

RFFE columns (sweep sequencer table)
The cells in the "RFFE" columns define the sequence of commands to be sent over the respective RFFE interface. The command definition dialog (opened from the ellipsis buttons) is described in "Basic RFFE interface settings and command execution" on page 677.

Remote command:
`CONTrol<Ch>:RFFE<Bus>:SETTings[:STATe]`
Unsegmented sweeps
`CONTrol<Ch>:SEQuence<SeqNr>:RFFE<Bus>:COMMand:DATA`
Segmented sweeps:
`CONTrol<Ch>:SEGMent<SegNr>:SEQuence<SeqNr>:RFFE<Bus>:COMMand:DATA`

GPIO columns (sweep sequencer table)
The cells in the "GPIO" columns define the sequence of voltages to be applied to the respective GPIO ports.

Remote command:
`CONTrol<Ch>:GPIO<Port>[:STATe]`
Unsegmented sweeps
`CONTrol<Ch>:SEQuence<SeqNr>:GPIO<Port>:VOLTage`
Segmented sweeps:
`CONTrol<Ch>:SEGMent<SegNr>:SEQuence<SeqNr>:GPIO<Port>:VOLTage`

Wait (Sweep Sequencer Table)
The cells in the "Wait" column define the delay times between subsequent sequence steps and between the final sequence step and the sweep (segment) start.

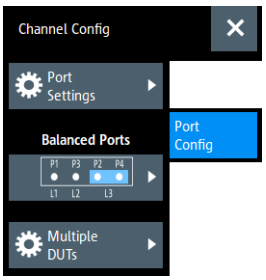
Remote command:
Unsegmented sweeps
`CONTrol<Ch>:SEQuence<SeqNr>:DELay`
Segmented sweeps:
`CONTrol<Ch>:SEGMent<SegNr>:SEQuence<SeqNr>:DELay`

5.12.2 Port Config tab

Configures the source and receive ports of the R&S ZNB, for measurements on frequency-converting DUTs, defining arbitrary port frequencies and powers. While the arbitrary mode is active, "Arb" and the selected stimulus axis are displayed in the channel line.

Background information
Refer to Section 4.7.3, "Frequency conversion measurements", on page 233.

5.12.2.1 Controls on the Port Config tab



If either multiple DUTs are configured (see Section 5.12.2.3, "Define Parallel Measurement dialog", on page 694) or the "Fixture Simulator" is disabled for the related channel (see "Fixture Simulator" on page 672), the "Balanced Ports" button is inactive (grayed out).

Port Settings
Opens a dialog that allows you to configure the source and receiver ports of the R&S ZNB for arbitrary frequencies, source and receiver levels: see Section 5.12.2.2, "Port Settings dialog", on page 686.

Balanced Ports
Opens a dialog that allows you to enable/disable physical ports and to define logical ports (balanced or unbalanced) in the active channel. See "Balanced Ports dialog" on page 296.

Note: If either multiple DUTs are configured (see Section 5.12.2.3, "Define Parallel Measurement dialog", on page 694) or the "Fixture Simulator" is disabled for the rela-

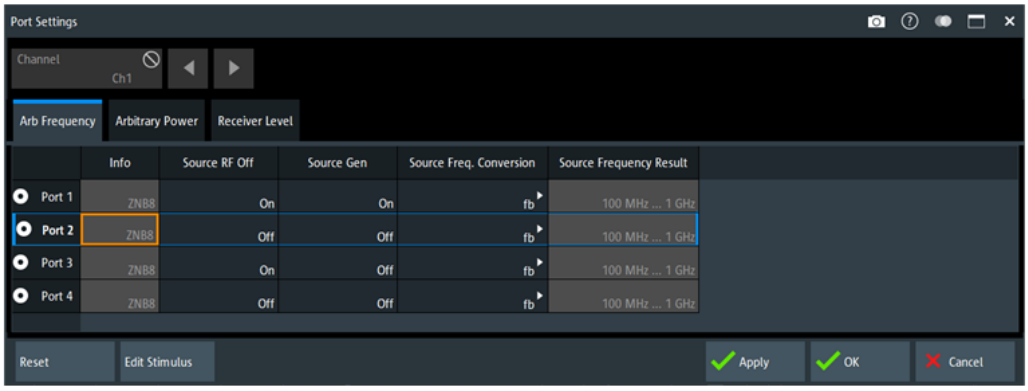
ted channel (see "Fixture Simulator" on page 672), the "Balanced Ports" button is inactive (grayed out).

Multiple DUTs
Opens a dialog that allows you to set up the analyzer for parallel measurements on multiple DUTs. See Section 5.12.2.3, "Define Parallel Measurement dialog", on page 694.

5.12.2.2 Port Settings dialog

The "Port Settings" dialog is an example of a Multi-channel setup dialog. For the selected channel, it configures the source and receiver ports of the R&S ZNB for arbitrary frequencies, source and receiver levels.

Access: Channel – [Channel Config] > "Port Config" > "Port Settings"



Related information
Refer to the following sections:

- Section 4.7.3, "Frequency conversion measurements", on page 233
- Section 5.17.4.2, "External Power Meters dialog", on page 845
- Section 5.17.5.2, "External Generators dialog", on page 851

Arb Frequency tab

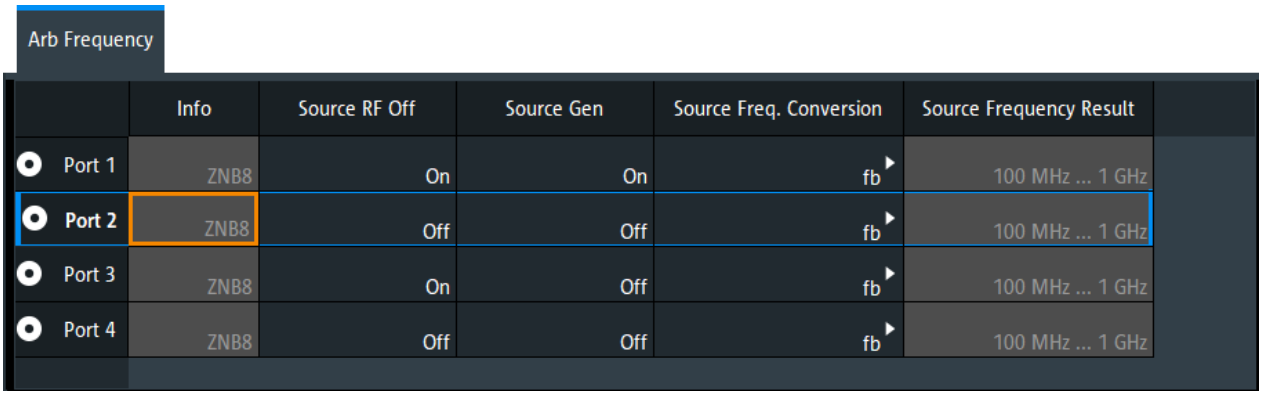
Defines port-specific frequencies (arbitrary mode) and other port-specific source settings.

Arbitrary mode requires software option R&S ZNB-K4, see Section 4.7.3, "Frequency conversion measurements", on page 233.

If any of the port frequencies are changed from their preset values, the frequency conversion mode is activated on pressing "Apply" or "OK" in the dialog. An "Arb" label appears in the channel line.

Channel base frequency f_b

For frequency sweeps, the channel base frequency f_b corresponds to the unmodified sweep range. For power, time, and CW mode sweeps, it is equal to the CW frequency. f_b is defined by the "Stimulus" parameters (see Section 5.8, "Stimulus softtool", on page 526).



Non-editable table cells

In addition to the test ports ("Port"), the source ports include all configured external generators ("Source Gen").

The receive ports include all configured external power meters ("Pmtr" ...).

Each port is displayed with its port number, device type ("Info"), and "Frequency Result", according to the current channel base frequency and "Source Freq. Conversion" settings.

Values that cannot be modified are displayed with a gray background; frequency results that cannot be measured are displayed with a red background.

Source RF

Turns the RF signal at a source port (analyzer test port or generator port) on or off.

While the RF power is switched off, the port can still be used as a receive port.

Remote command:

```
SOURce<Ch>:POWer<PhyPt>:STATe
SOURce<Ch>:POWer:GENerator<Gen>:STATe
```

Source Gen

Defines the behavior of the RF signal source at the related port.

- If disabled (default), it is only switched on for the partial measurements that require the port as a drive port.
- If enabled, it is switched on for all partial measurements (permanent signal source).

"Source Gen" has no effect, if Source RF is enabled.

Permanent signal sources can eliminate the power settling times of the DUT but can introduce measurement inaccuracies, e.g. due to crosstalk between ports. Therefore, during system error corrections permanent signal sources must be deactivated.

Note:

- For the R&S ZNB, among the analyzer ports driven by the same internal source, only one can operate as generator port.

An **external generator** always represents a permanent signal source that is switched on for all partial measurements. "Source Gen" determines whether the generator is available as a signal source in the test setup. The analyzer provides two alternative, independent ways of selecting a generator as a signal source:

- Enable "Source Gen" in the "Port Settings" dialog, especially if the generator is not assigned to a particular measurement result or drive port.
- Select the generator as a source for a particular measurement result or for a power calibration, see e.g. "More Wave Quantities dialog" on page 307.

Note: If external switch matrices are used, only external generators can serve as generator ports. The "Source Gen" flag for internal ports is unset and disabled.

Remote command:

```
SOURce<Ch>:POWer<PhyPt>:PERManent[:STATe]
SOURce<Ch>:POWer<PhyPt>:GENerator<Gen>:PERManent[:STATe]
```

Source Freq. Conversion

Opens a dialog to define a port-specific stimulus frequency range (for frequency sweeps) or CW frequency (for power, time and CW mode sweeps); see "Modify Frequency Conversion dialog" on page 688. In the default configuration, the channel base frequency f_b is used.

This cell is only enabled, if the analyzer is equipped with option R&S ZNB-K4.

The result is displayed in the "Frequency Result" column. A red "Source Freq. Conversion" cell indicates that certain hardware limits are exceeded.

Remote command:

```
SOURce<Ch>:FREQuency<PhyPt>:CONVersion:ARBitrary:IFRequency
```

Frequency Result

Displays the resulting receiver frequencies.

Modify Frequency Conversion dialog

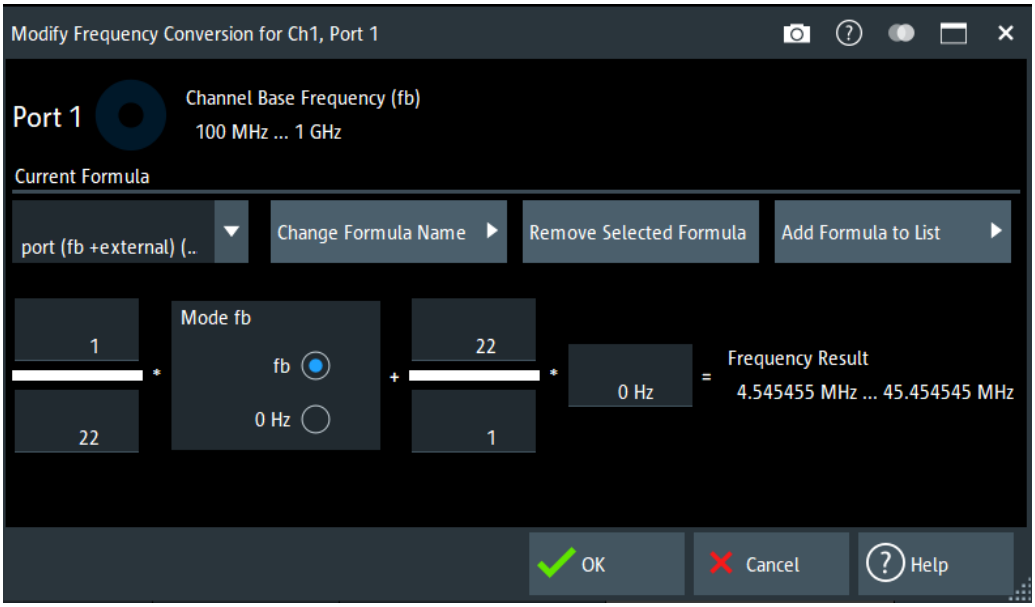
The "Modify Frequency Conversion" dialog defines a port-specific frequency formula (for frequency sweeps) or CW frequency (for power, time and CW Mode sweeps).

Access: Port Settings dialog > "Arb Frequency" tab > "Source Freq. Conversion" cells

Dependencies

The analyzer firmware enforces the following rules:

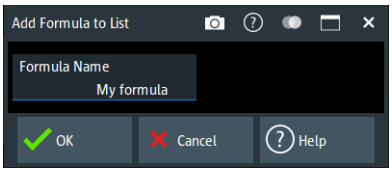
- For balanced ports, the constituent physical ports must use the same Frequency Conversion Formula.
- Source Gen ports driven by the same internal source (R&S ZNBT only) must use the same frequency conversion formula.



Current Formula

Apart from the current Frequency Conversion Formula, the analyzer firmware maintains a list of named, user-defined formulas.

- "Add Formula to List..." adds the current frequency conversion formula to this list.



- "Remove Selected Formula" removes the selected user-defined formula from this list.
- "Change Formula Name..." allows you to change the name of the currently selected user-defined formula.

Note:

- User-defined formulas are persisted in the recall set. A preset deletes all user-defined formulas.
- User-defined formulas are updated automatically: if a user-defined formula is selected and the "Frequency Conversion Formula" on page 689 is modified, then the formula itself is modified. As a consequence, all "Source Freq. Conversion" definitions using this formula are modified as well.
- If a user-defined frequency conversion formula is removed from the list, all "Source Freq. Conversion" definitions using this formula are replaced by the "unlinked" formula.

Frequency Conversion Formula

The conversion formula has been modeled according to the needs of a typical frequency-converting measurement where mixer products, intermodulation products, or harmonics occur.



- For mixer measurements, select a "Base Ratio" of 1, a "Frequency Offset" equal to the LO frequency, and an "Offset Ratio" of ±1. If the RF signal is at the channel base frequency, the port frequency is at the frequency of the upper (lower) sideband.
- To measure n-th order intermodulation products, select integer "Base Ratio" and "Offset Ratio" with |"Base Ratio"| + |"Offset Ratio"| = n.
- To measure the n-th harmonic of the channel base frequency, select a "Frequency Offset" of 0 and an integer "Base Ratio" = n.

Remote command:

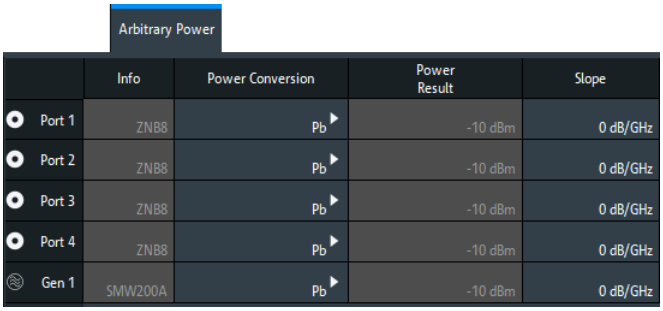
```
SOURce<Ch>:FREQuency<PhyPt>:CONVersion:ARBitrary:IFRequency
SOURce<Ch>:FREQuency<PhyPt>:CONVersion:ARBitrary:EFRequency<Gen>
[SENSe<Ch>:]FREQuency:CONVersion:ARBitrary:PMETer<Pmtr>
```

Arbitrary Power tab

Configures the source ports of the R&S ZNB for port-specific powers.

Channel base power P_b

For power sweeps, the channel base power P_b corresponds to the unmodified sweep range. For frequency, time, and CW mode sweeps, the channel base power P_b is equal to the fixed "Power". f_b is defined by the "Stimulus" parameters (see Section 5.8, "Stimulus softtool", on page 526).



Non-editable table columns

In addition to the test ports ("Port"), the source ports include all configured external generators ("Source Gen"). Each port is displayed with its port number and device type ("Info").

For each port, the available power range and resulting port power are calculated from the configured power conversion and source attenuation settings.

Power Conversion
Opens a dialog that allows you to define a port-specific source power range (for power sweeps) or fixed power (for frequency, time and CW mode sweeps) for the current channel. See "Cal Power Config dialog" on page 691.

In the default configuration, the channel base power P_b is used. The frequency-adjusted result is displayed in the "Power Result" column.

Slope
Defines a linear factor to modify the port-specific source power as a function of the stimulus frequency. The resulting output power at port frequency f is calculated as <Power Result> + <Slope> * f.

Remote command:
`SOURce<Ch>:POWer<PhyPt>[:LEVel][:IMMediate]:SLOPe`
Generator: `SOURce<Ch>:POWer:GENerator<Gen>:SLOPe`

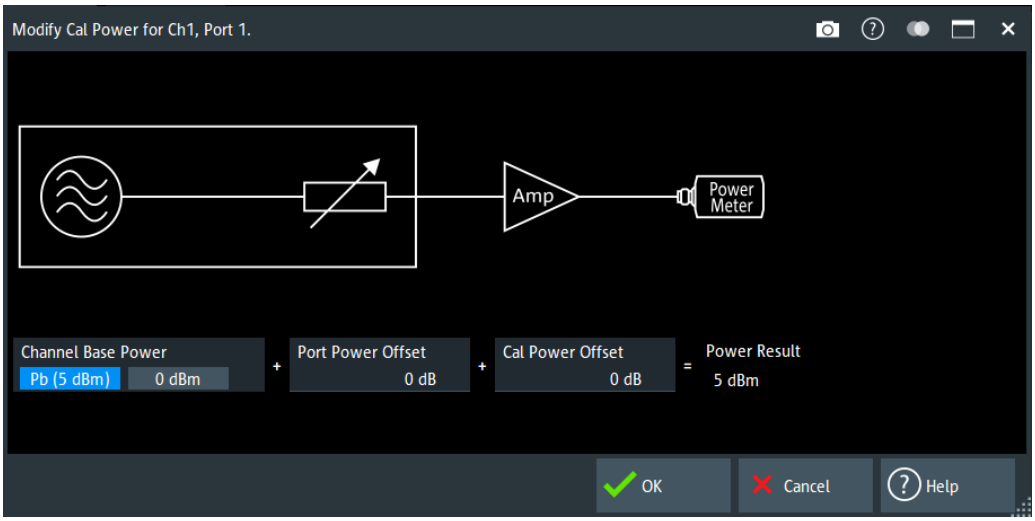
Cal Power Config dialog

The "Cal Power Config" dialog allows you to define port-specific source powers, which can be necessary to ensure a definite signal power at the inputs of the DUT.

Access: Port Settings dialog > "Arbitrary Power" tab > "Power Conversion" cells

Related information
The settings in the "Cal Power Config" dialog are also used to define the target power for the scalar power calibration; refer to Section 5.11.3.2, "Cal Power Config dialog", on page 650.

The diagram in the center of the dialog visualizes the settings and results below.



The control elements in the "Cal Power Config" dialog are described in the following sections:

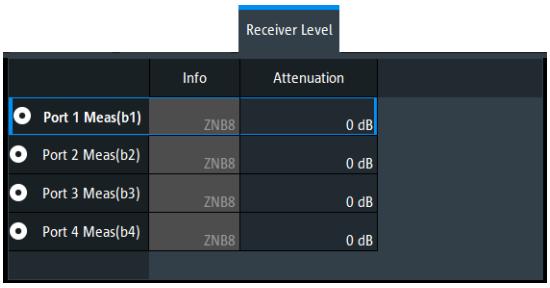
- "Channel Base Power" on page 651
- "Port Power Offset" on page 651

- "Cal Power Offset" on page 652

Receiver Level tab

Provides access to the receiver step attenuator settings.

Receiver step attenuators are optional hardware (see Section 4.7.24, "Receiver step attenuators", on page 264).



Attenuation
Sets the attenuation at the respective measurement receiver (b-wave). See "Step Attenuators" on page 532.

Remote command:
`[SENSe<Ch>:]POWer:ATTenuation`

Common controls

Depending on the available options and the setup, some parameters may be hidden.

Channel
Shows the channel for which the port settings are applied with the "Apply" and "OK" buttons.

Previous / Next
If several channels have been configured, you can change the channel for which the port settings are applied with the "Apply" and "OK" buttons.

Edit Stimulus
For non-segmented sweeps, the "Edit Stimulus..." button opens the Stimulus dialog that allows you to access the stimulus settings without having to leave the "Port Settings" dialog.

Apply / OK / Cancel
Modified "Port Settings" take effect on "Apply" or "OK". Use "Cancel" to discard possible changes.

Remote command:
`[SENSe<Ch>:]FREQuency:CONVersion ARBitrary | FUNDamental`

Stimulus dialog

The "Stimulus" dialog gives access to the parameters of a non-segmented sweep.

Access: Port Settings dialog > "Edit Stimulus..."

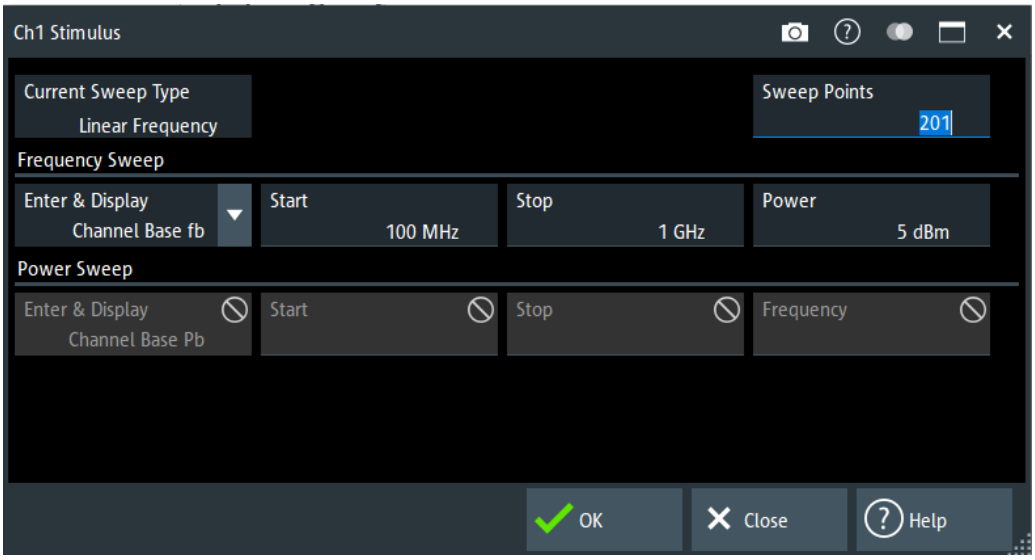


Figure 5-70: Stimulus Dialog: Linear Frequency Sweep

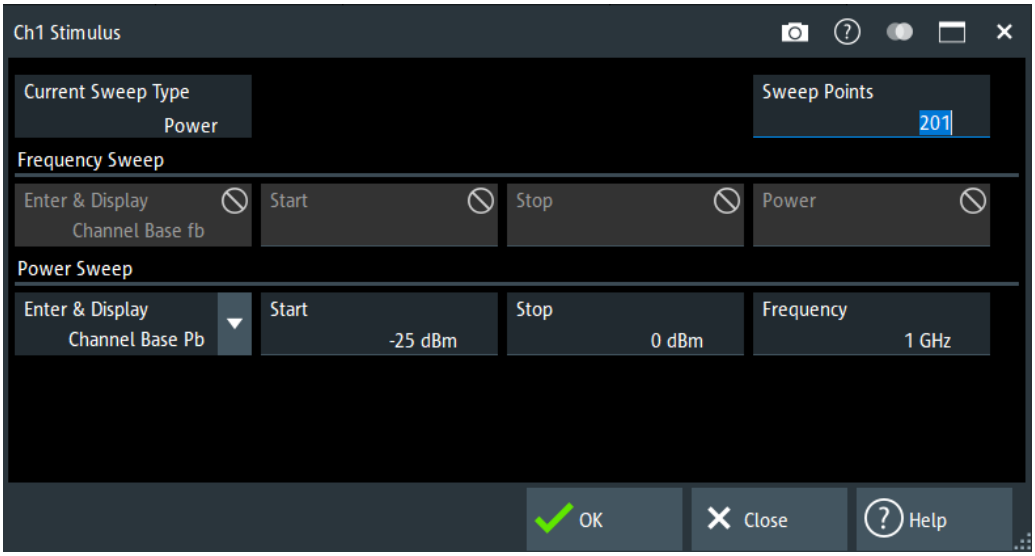


Figure 5-71: Stimulus Dialog: Power Sweep

Except for the "Enter & Display" properties, all these parameters can also be set in the Sweep Softtool. Depending on the channel's Sweep Type, a different set of parameters can be modified.

Table 5-6: Parameters for frequency sweeps

	Lin/Log Freq Sweep	CW Mode Sweep	Time Sweep
Enter & Display	x	x	x
Start	x	–	–
Stop	x	–	–
Power	x	x	x
Number of Sweep Points	x	x	x

Enter & Display
The "Enter & Display" combo boxes in the "Frequency Sweep" and "Power Sweep" sections of the "Stimulus" dialog control how frequency and power values are entered and displayed in this dialog.

You can either take the channel perspective (f_b, P_b), or the perspective of a particular source or receiver. The two can be used interchangeably to define the sweep, if the user-defined linear transformation relating them is not constant.

Note: For frequency sweeps, source/receiver-specific linear frequency transformations are enabled with option R&S ZNB-K4 (see Section 4.7.3, "Frequency conversion measurements", on page 233).

5.12.2.3 Define Parallel Measurement dialog

With the R&S ZNBT or a 4-port R&S ZNB with Second internal generator, you can measure multiple DUTs (or multiple paths of a single DUT) in parallel.

The "Define Parallel Measurement" dialog allows you to configure these DUTs and to declare the connections between physical VNA ports and DUT ports. Furthermore it provides convenient access to port-related settings.

Access: Channel – [Channel Config] > "Port Config" > "Multiple DUTs"

Related information
See Section 4.1.4.3, "Parallel measurements on multiple DUTs", on page 85.

For the R&S ZNB, the "Parallel Measurement" is automatically configured when "Enabled" is selected, as shown in Figure 5-72.

A R&S ZNB with two internal sources allows parallel measurements with "Frequency Offset", which can be configured in the corresponding tab on the right.

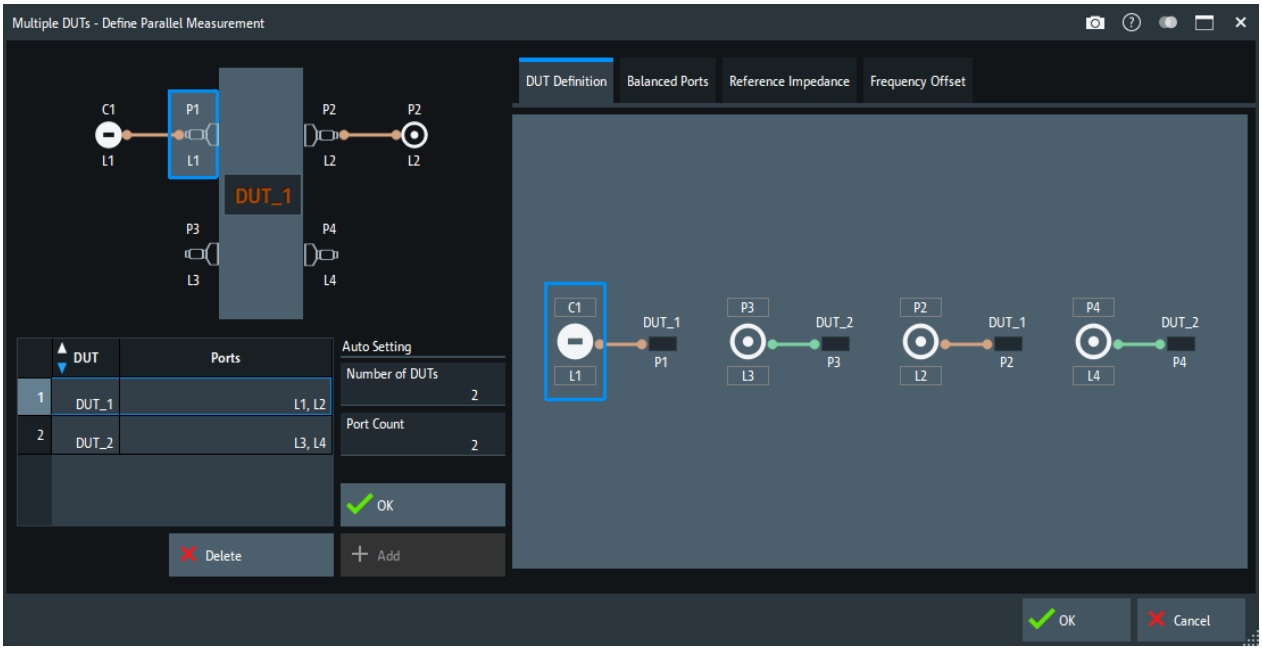


Figure 5-72: Define Parallel Measurement (R&S ZNB)

- Modified settings are not applied unless the "Define Parallel Measurement" dialog is closed with the "OK" button.
- On "OK", any pre-existing logical port configuration is overwritten.
- Each configured DUT can have its own ground loop embedding and deembedding network

DUT Definition tab

Modify an existing DUT

The DUT table allows you to rename an existing DUT (R&S ZNB and R&S ZNBT) and to define its physical ports and drive port order (R&S ZNBT only).

1. Locate the related row in the DUT table.
2. In the "DUT" column, specify the DUT's name.

Controls and Functions

The GUI functions are self-explanatory; their use is sufficiently explained in the above procedures.

The remote control implementation of the "Multiple DUTs" feature introduces an additional "port group" layer. A DUT's port group consists of the logical VNA ports that are connected to this DUT.

Remote command:

```
SOURce<Ch>:GROup<Grp>:PPORts
SOURce<Ch>:GROup:COUNt?
```

```
SOURce<Ch>:GROup<Grp>:NAME
SOURce<Ch>:GROup<Grp>:DPORt:COUNt
SOURce<Ch>:GROup<Grp>:PPORt<PhyPort>:DPORt
```

Balanced Ports tab

The "Balanced Ports" tab of the "Define Parallel Measurement" dialog allows you to define the balanced ports of the configured DUTs.

To modify the balanced port configuration, proceed as follows:

1. Select the related row in the DUT table.

   The DUT perspective (upper left part of the dialog) now displays the selected DUT.
2. Activate the "Balanced Ports" tab.
3. To create a balanced port:
   a) In the DUT perspective, toggle select two DUT ports.
   b) In the "Balanced Ports" tab, select "Balanced" to create the balanced port from the selected DUT ports.
4. To dissolve one or more balanced ports:
   a) In the DUT perspective, for each balanced port you want to dissolve, toggle-select one of its constituent DUT ports.
   b) In the "Balanced Ports" tab, select "Single" to dissolve all related balanced ports.

Reference Impedance tab

The "Reference Impedance" tab of the "Define Parallel Measurement" dialog allows you to set the reference impedance for balanced and unbalanced DUT ports.

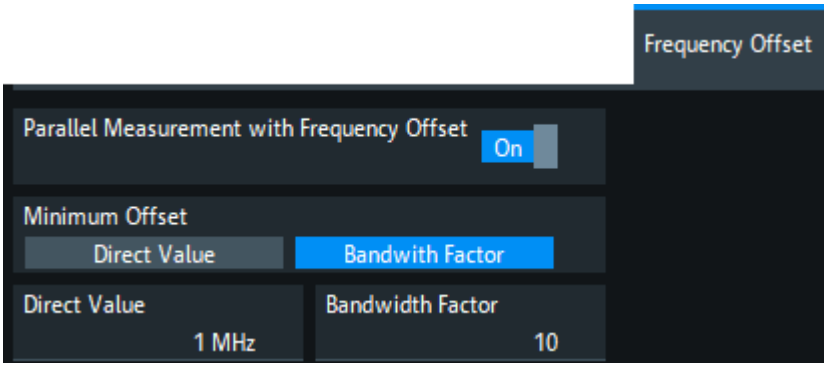
To change the reference impedance for a particular DUT port, proceed as follows:

1. Select the related row in the DUT table (lower left part of the dialog).

   The DUT perspective (upper left part of the dialog) now displays the selected DUT.
2. Activate the "Reference Impedance" tab.
3. In the DUT perspective, toggle select (one of) the related DUT ports.
4. Use the controls in the Reference Impedance tab to specify the impedance settings. The user interface is identical to the one described in "Reference Impedance tab" on page 300.

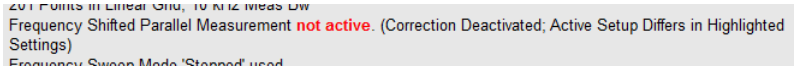
Frequency Offset tab

The "Frequency Offset" tab allows you to activate Parallel measurements on multiple DUTs and to define a "Minimum Offset" between the stimuli.



To get access to the "Frequency Offset" configuration, create multiple DUTs (on the R&S ZNB, set "Parallel Measurement" to "Enabled") and activate the "Frequency Offset" tab.

It is essential to perform the calibration with the same "Frequency Offset" settings as for the actual measurement; otherwise the calibration is deactivated ("Cal Off"). If there is a mismatch, the Calibration Manager dialog provides additional information.



Parallel Measurement with Frequency Offset
Enables/disables a frequency offset in parallel measurements.

For segmented sweeps, the parallel measurements are performed segment per segment. If the Minimum Offset is specified as a "Bandwidth Factor" and the segments use different measurement bandwidths, then the resulting frequency offsets are also different.

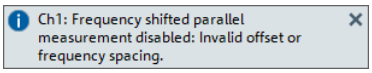
Note: If both "Parallel Measurement with Frequency Offset" and AGC Mode "Auto" are active, automatic switches between small and large IF gain during the sweep can lead to visible steps in the measurement results. Select a fixed AGC mode ("Low Dist" or "Low Noise") to suppress this effect.

Remote command:
`SOURce<Ch>:GROup<Grp>:SIMultaneous:FOFFset:STATe`

Minimum Offset
The "Minimum Offset" can be specified either as an absolute value ("Direct Value") or as a multiple of the measurement bandwidth ("Bandwidth Factor").

Remote command:
`SOURce<Ch>:GROup<Grp>:SIMultaneous:FOFFset:MOFFset:MODE`
`SOURce<Ch>:GROup<Grp>:SIMultaneous:FOFFset:MOFFset:DVALue`
`SOURce<Ch>:GROup<Grp>:SIMultaneous:FOFFset:MOFFset:BWFactor`

State indication warning
When attempting to activate the "Parallel Measurement with Frequency Offset" (i.e. on closing the "Define Parallel Measurement" dialog with the "OK" button), the VNA Firmware checks whether the requested settings are technically feasible. If not, the firmware disables parallel measurement (i.e. the measurements are performed one after the other) and generates a warning message indicating the current configuration problem.



In particular, activation is rejected if:
- The start frequency is < 31 MHz
- The measurement bandwidth is > 100 kHz
- The resulting frequency offset (i.e. the minimum frequency offset, rounded to a multiple of the current frequency step size) would be too high
- The port groups are configured with different frequency conversion settings

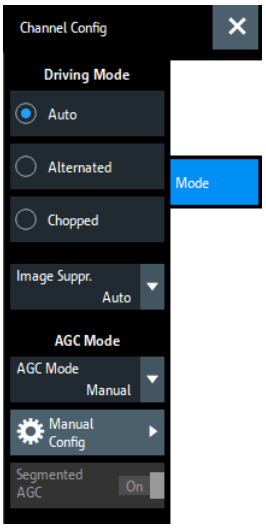
The generated error code can be retrieved via remote control command `SOURce<Ch>:GROup:SIMultaneous:FOFFset:CONDition?`. The configuration problems listed above are indicated as error codes -8 "invalid offset or frequency spacing" and -6 "no simultaneous mode possible".

Remote command:
`SOURce<Ch>:GROup<Grp>:SIMultaneous:FOFFset:CONDition`

5.12.3 Mode tab

Optimizes the measurement process.

5.12.3.1 Controls on the Mode tab



The controls on the "Mode" tab are organized in sections "Driving Mode", "Image Suppr." and "AGC Mode".

Driving Mode
Determines the order of partial measurements and sweeps.

- In "Auto" mode (default), the analyzer optimizes the display update: Fast sweeps are performed in "Alternated" mode, slower sweeps in "Chopped" mode.
- In "Alternated" mode, a partial measurement is performed at all sweep points (partial sweep) before the hardware settings are changed. The next partial measurement is carried out in an additional sweep.
  This mode is usually faster than "Chopped" mode.
- In "Chopped" mode, the analyzer completes the necessary sequence of partial measurements at each sweep point and obtains the result (measurement point) before proceeding to the next sweep point. A trace is obtained from the beginning of the sweep.

The "Driving Mode" setting is also used during a system error correction. For channels which require a single partial measurement only, the driving mode settings are equivalent. See also Section 4.1.4.1, "Partial measurements and driving mode", on page 83.

Note: The "Block Alternated" mode and the related setting "Number of Alternated Blocks" were removed with FW V4.0.

Remote command:
`[SENSe<Ch>:]COUPle`

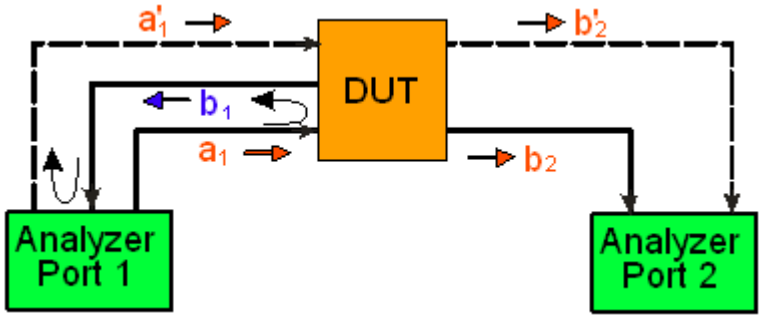
Image Suppr.
The "Image Suppr." settings define whether the analyzer measures with a local oscillator frequency LO below or above the RF input frequency. This feature can be used to eliminate known spurious components in the input signal that can distort the measurement, especially in the low frequency range.

- In "Auto" mode, the analyzer auto-selects the LO frequency, depending on the receiver (RF) frequency and the test port. This mode systematically avoids known spurious signals if no frequency conversion occurs in the test setup.
- "LO < RF" means that the LO frequency is always below the measured RF frequency. This mode is appropriate for avoiding single, known spurious signals.
- "LO > RF" means that the LO frequency is always above the measured RF frequency. This mode is appropriate for avoiding single, known spurious signals.

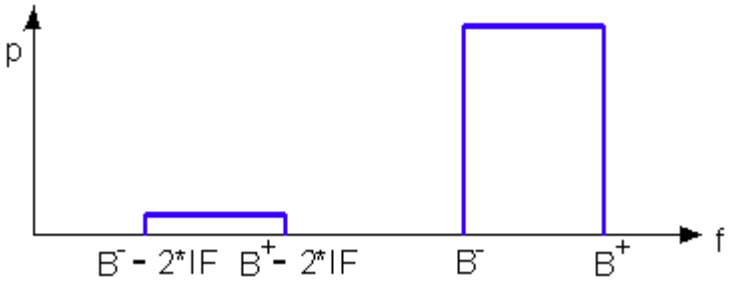
Tip: In the presence of several spurious signals, setting the "Image Suppr." parameter globally can be insufficient. To improve the result, perform a segmented frequency sweep and assign independent LO frequencies to the individual sweep segments.

Application example

Consider the following test setup with a strongly reflecting DUT (e.g. a bandpass in its stop band) that is measured in transmission. The incident wave a_1 is generated at a frequency RF. The reflected wave b_1 falls into the receiver mixer of the analyzer port 1; a small fraction of the mixer product RF + 2*IF can be reflected back towards the DUT. If this spurious wave a'_1 passes the DUT, then it is received as b'_2 at port 2, together with the wanted signal b_2.



LO > RF implies that LO = RF + IF. The mixer at port 2 converts both the wanted signal b_2 and the spurious signal b'_2 which is at the frequency RF' = IF + LO, to the same IF frequency. The response of an ideal, infinitely steep bandpass filter with a pass band between B^- and B^+ looks as follows:



For a wide bandpass, the spurious response flattens the filter edges.

The spurious signal can be eliminated by dividing the sweep range into two segments with different LO settings:

- In the low-frequency segment ranging up to the center frequency of the bandpass filter, the frequency of the local oscillator is set to LO < RF. This setting ensures that the spurious signal b'_2 is not measured at port 2.
- In the high-frequency segment, starting at the center frequency of the bandpass filter, the frequency of the local oscillator is set to LO > RF. If the center frequency is larger than $B^+ – 2*IF$, then there is no distortion from b'_2.

Remote command:
`[SENSe<Ch>:]FREQuency:SBANd`

LO Suppression
A small portion of the local oscillator (LO) driving the mixer of the receivers, can leak out of the test ports of the R&S ZNB. This power leakage can lead to compression of high-gain DUTs.

"LO Suppression" lowers the LO power of all receivers and thus the power leaking out of the test ports.

R&S ZNB with recent synthesizer boards perform "LO Suppression" automatically and hence this control is hidden.

Remote command:
`[SENSe<Ch>:]POWer:LOSuppress[:STATe]`

AGC Mode
Configures the gain control (GC) logic.

In **"Auto"** mode, the analyzer adapts its receiver step attenuator settings to the RF input level (→ Automatic or Adaptive Gain Control, AGC). The A/D converter is always operated at optimum input level, selecting one of the following gain settings for every measurement point:

- **"Low Noise"** corresponding to a higher internal A/D converter input level. This setting increases the dynamic range and is appropriate for low RF input levels.
- **"Low Dist"**, corresponding to a lower internal A/D converter input level. This setting allows for a high RF overdrive reserve and is appropriate for high RF input levels.

"Low Dist" and "Low Noise" can also be selected **statically**, completely disabling the adaptive behavior. A static GC mode is appropriate, if the characteristics of the input paths must be constant, e.g. because:

- interfering signal contributions originating from the receiver (noise, nonlinear contributions) must not change during the measurement.
- a large interfering signal close to the measured signal must not overdrive the receiver.

"Manual" mode allows you to select the preferred "AGC Mode" per sweep segment, drive port and receiver (see "Drive-port specific settings" on page 702).

Remote command:
`[SENSe<Ch>:]POWer:GAINcontrol:GLOBal`

Manual Config
The "Manual Config" button opens the AGC Manual Configuration dialog that allows to configure the GC for the individual sweep segments, drive ports and receivers. This button is enabled in "Manual" AGC Mode only.

Segmented AGC
"Segmented AGC" enables segment-specific gain control settings. It is available for "Segmented" sweep type only.

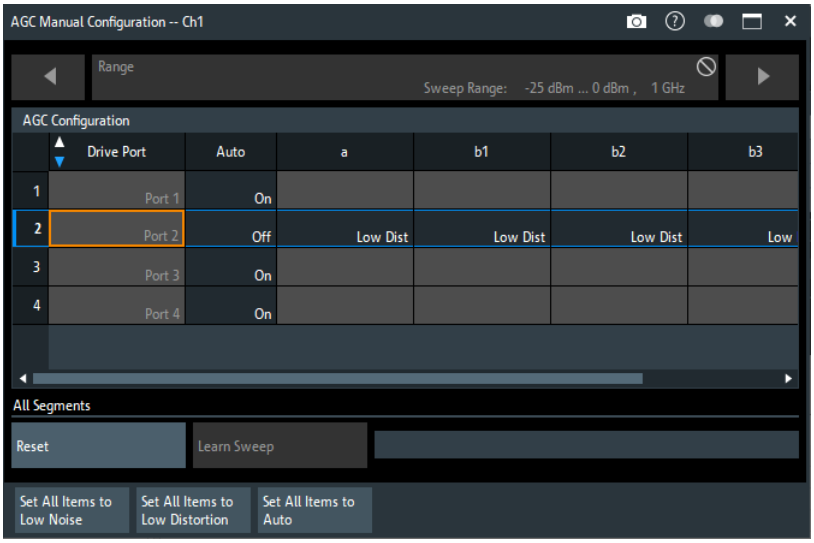
Remote command:
`[SENSe<Ch>:]SEGMent<Seg>:POWer:GAINcontrol:CONTrol`

5.12.3.2 AGC Manual Configuration dialog

The "AGC Manual Configuration" dialog allows you to configure the AGC settings for individual sweep segments, drive ports and receivers. This button is enabled in "Manual" AGC Mode only.

Access: Channel – [Channel Config] > "Mode" > "AGC Mode" – "Manual Config"

The manual AGC settings are persisted and reused for subsequent measurement sweeps, which can increase measurement speed compared to "Auto" mode while not compromising measurement quality.



- Without the Extended power range option only the AGC of the measurement receivers (the b-waves) can be statically set to "Low Dist" or "Low Noise"; the reference receiver AGC mode is always set to "Auto" (see "AGC Mode" on page 701). With the option available, also the AGC of the reference receivers (the a-waves) can be set statically.
- The increase in measurement speed for settings "Low Dist" and "Low Noise" is not achieved if the AGC mode of one of the receivers is set to "Auto".

Range
If Segmented AGC is enabled, each sweep segment can be configured separately.

Remote command:
`[SENSe<Ch>:]SEGMent<Seg>:POWer:GAINcontrol`
`[SENSe<Ch>:]SEGMent<Seg>:POWer:GAINcontrol:ALL`

Drive-port specific settings
"Drive Port", "a", "b<j>": Selects the AGC mode for the respective drive port, a and b wave and receivers.

"Auto" (column): Enables the automatic mode for the corresponding drive port, disabling the manual configuration for the related a- and b-waves.

Remote command:
`[SENSe<Ch>:]POWer:GAINcontrol`

All Segments – Reset
Restores the default settings for all segments and drive ports.

All Segments – Learn Sweep
During a learn sweep, the analyzer determines the appropriate static gain settings for the measured a- and b-waves, i.e. for all a- and b-waves measured in the current channel. The acquired settings can be overwritten manually.

At the start of the learn sweep, "Auto" mode is selected for each drive port ("Reset") and a single shot measurement of the current channel is initiated.

During this measurement, the VNA firmware observes the "Low Dist" (LD) vs. "Low Noise" (LN) gain decisions of the AGC. It then determines the statically assigned gain for the individual sweep segments / drive ports / receivers in the following way:

- if LD was selected for any of the related measurement points, then LD is assigned
- otherwise LN is assigned

In other words, LN is assigned if and only if LN was selected for all related measurement points.

Note:

- Before running the "Learn Sweep", create the adequate port configuration and add the required traces.
- The "Learn Sweep" is not available for power sweep channels.

Remote command:

```
[SENSe<Ch>:]POWer:AGCMode:ACQuire
[SENSe<Ch>:]POWer:AGCMode:SAVE
```

Set All Items to ...

Convenience functions to apply the same manual AGC mode to all a and b wave receivers.

If Segmented AGC is enabled, the setting only applies to the selected segment.

Remote command:

```
[SENSe<Ch>:]POWer:GAINcontrol:ALL
```

5.12.4 Pwr Cal Settings tab

Replicates the "Pwr Cal Settings" tab of the "Cal" softtool (see Section 5.11.3, "Power Cal Settings tab", on page 648).

5.13 Offset Embed softtool

The "Offset Embed" softtool allows you to define a length offset and loss for each test port. The offset compensates for the known length and loss of (non-dispersive and perfectly matched) transmission lines between the calibrated reference plane and the DUT. It also contains advanced functions for deembedding/embedding the DUT from/into more general physical/virtual (matching) networks placed between the calibrated reference plane and the DUT.

Access: Channel – [Offset Embed]

Background information

Refer to Section 4.6, "Offset parameters and de-/embedding", on page 197

5.13.1 Offset Embed dock widget

On activating a tab in the "Offset Embed" softtool, a dock widget is displayed beneath the trace area, whose content pane is synchronized with the selected softtool tab.

Access: Channel – [Offset Embed]

5.13.1.1 Common controls

Overview
Opens the Overview panel. This button is available on all "Offset Embed" dock widget panels – except the "Overview" panel itself.

Close
Closes the "Offset Embed" dock widget.

All Deembedding Activated
De-/activates all deembeddings (single-ended, balanced, port set, ground loop) configured for the active channel.

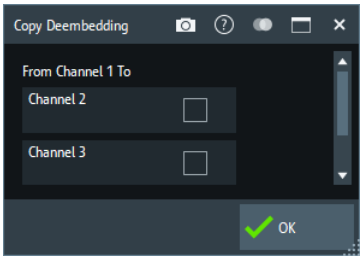
This control is only available on the Overview panel, where it replaces the Overview button.

Remote command:
`CALCulate<Ch>:TRANsform:VNETworks:ACTivateall[:STATe]`

Copy Deembedding to Other Channels
This function is only available for deembedding types Single Ended, Port Sets, and Balanced. It is only enabled if the current setup contains more than one channel.

Using this function you can copy the respective deembedding settings from the active channel to other channels.

The "Copy Deembedding to Other Channels" button opens a dialog that allows you to select the target channels.



Remote command:
`CALCulate:FMODel:COPY:CHANnel:DEEMbedding:SINGleended`
`CALCulate:FMODel:COPY:CHANnel:DEEMbedding:PSET`
`CALCulate:FMODel:COPY:CHANnel:DEEMbedding:BALanced`

5.13.1.2 Overview panel

Shows an overview of the overall calculation flow and provides quick access to the "Offset Embed" functions.

Access: Channel – [Offset Embed] > "Overview"

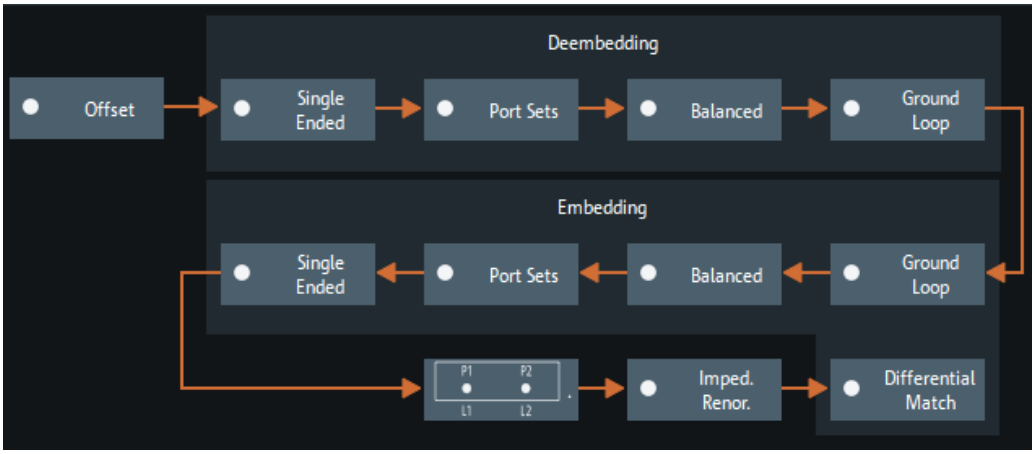


Figure 5-73: Offset Embed dock widget: Overview (Calculation Flow)

Use one of the buttons to configure the corresponding function. A green LED on a button indicates that the corresponding deembedding/embedding function is active.

If the "Fixture Simulator" is disabled for the related channel (see "Fixture Simulator" on page 672), all functions except the "Offset" function are in inactive (grayed out in the "Overview").

5.13.1.3 Offset panel

Allows you to activate or deactivate offset/loss correction for selected physical ports and to set related parameters.

Access: Overview panel > "Offset"

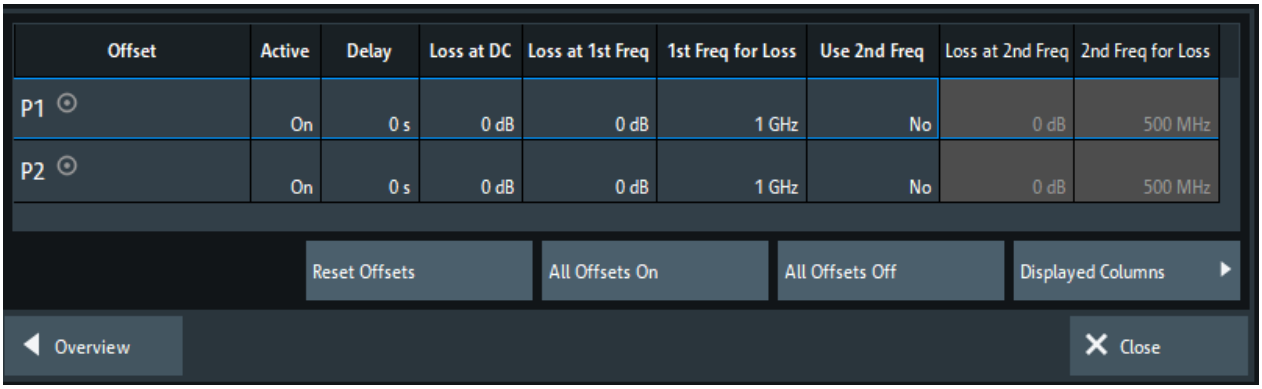


Figure 5-74: Offset Embed dock widget: Offset panel

The "Offset" panel can also be activated by selecting the Offset tab or One Way Loss tab. Refer to its description for additional functions and remote commands.

Active
The control in the "Active" column enables or disables the configured length and loss parameters for the respective Port (i.e. adds/removes them to/from the calculation flow) without changing the parameter values.

Remote command:
`[SENSe<Ch>:]CORRection:OFFSet<PhyPt>:COMPensation[:STATe]`

Delay
See "Delay / Electrical Length / Mech. Length" on page 738.

Loss at DC / Loss at 1st Freq / 1st Freq for Loss / Use 2nd Freq / Loss at 2nd Freq / 2nd Freq for Loss
Defines the one-way loss parameters of the transmission line at the selected port.

In the limit case, where the length of the transmission line is considered to be "almost zero", the loss is considered constant:

$$Loss(f) = Loss_{DC}$$

Otherwise, if the loss at DC and one additional frequency f_1 is known, the loss at frequency f is approximated by:

$$Loss(f) = Loss_{DC} + (Loss(f_1) - Loss_{DC}) \cdot \sqrt{\frac{f}{f_1}}$$

If, in addition, the loss at a second frequency f_2 is known and "Use 2nd Freq" is enabled, then the loss is approximated by:

$$Loss(f) = Loss_{DC} + (Loss(f_1) - Loss_{DC}) \cdot \left(\frac{f}{f_1}\right)^b \text{ , where } b = \frac{\log_{10}\left(\left|\frac{Loss(f_1)-Loss_{DC}}{Loss(f_2)-Loss_{DC}}\right|\right)}{\log_{10}\left(\frac{f_1}{f_2}\right)}$$

See also Section 4.6.1.2, "Definition of loss parameters", on page 198.

Note:
- For a delay (or el./mech. length) of 0, the losses at frequency 1 and 2 are **not** applied.

The entered parameters define the loss for a signal traveling in **one direction** through the transmission line. To account for the propagation in both directions, the magnitude shift of a reflection parameter due to a given loss is twice the magnitude shift of a transmission parameter. See also Section 4.6.1.6, "Application and effect of offset parameters", on page 202.

Remote command:
`[SENSe<Ch>:]CORRection:LOSS<PhyPt>:OFFSet`
`[SENSe<Ch>:]CORRection:LOSS<PhyPt>`
`[SENSe<Ch>:]CORRection:LOSS<PhyPt>:FREQuency`
`[SENSe<Ch>:]CORRection:LOSS<PhyPt>:SECond:STATe`

```
[SENSe<Ch>:]CORRection:LOSS<PhyPt>:SECond
[SENSe<Ch>:]CORRection:LOSS<PhyPt>:SECond:FREQuency
```

Reset Offsets
The "Reset Offsets" button resets the length and loss parameters for all ports to their default values.

Remote command:

```
[SENSe<Ch>:]CORRection:OFFSet<PhyPt>[:STATe]
```

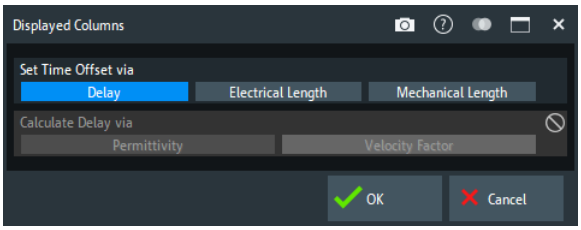
All Offsets On / All Offsets Off
Enables or disables the length and loss compensation for all ports.

Use the controls in the Active column to activate/deactivate the length and loss compensation for selected ports.

Remote command:

```
[SENSe<Ch>:]CORRection:OFFSet<PhyPt>:COMPensation[:STATe]
```

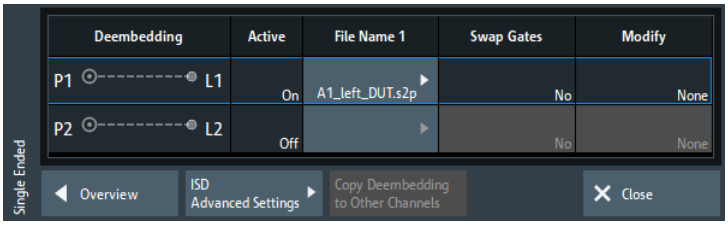
Displayed Columns
Opens a dialog that lets you select between different ways to specify the offset, similar to Delay / Electrical Length / Mech. Length on the Offset tab.



5.13.1.4 Single Ended panel

Allows you to activate or deactivate single ended deembedding/embedding for selected physical ports. For the "2-Port Data" network type, it is also possible to change the underlying `s2p` Touchstone file from the dock widget.

Access: Overview panel > "Single Ended"



The "Single Ended" panel can also be activated by selecting the Single Ended tab softtool tab. Refer to its description for background information, parameters and additional remote commands.

Active
The control in the "Active" column enables or disables the selected "Single Ended" de-/embedding Single Ended tab (i.e. adds or removes it to/from the calculation flow) without changing its parameters.

Remote command:
```
CALCulate<Ch>:TRANsform:VNETworks:SENDed:DEEMbedding<PhyPt>[:STATe]
CALCulate<Ch>:TRANsform:VNETworks:SENDed:EMBedding<PhyPt>[:STATe]
```

File Name 1/Swap Gates
The "2-Port Data" network is defined by its S-parameters stored in a two-port Touchstone file (`*.s2p`). No additional parameters are required. If you select this network for the respective port, the corresponding "File Name 1" cell becomes enabled.

A click or tap opens the Open de-/embedding file dialog.

Note: The loaded file is stored in the active recall set. Recall sets contain the full de-/embedding data so that they can be transferred to other instruments.

For "Swap Gates", see "Increasing Port Sequence/Swap Gates" on page 759.

Remote command:
```
MMEMory:LOAD:VNETworks<Ch>:SENDed:DEEMbedding<PhyPt>
MMEMory:LOAD:VNETworks<Ch>:SENDed:EMBedding<PhyPt>
```

Modify
For the "2-Port Data" network, you can decide to ignore DUT reflections by selecting "Set Snn @ DUT = 0".

This can significantly improve the accuracy of through measurements when using high-loss test fixtures.

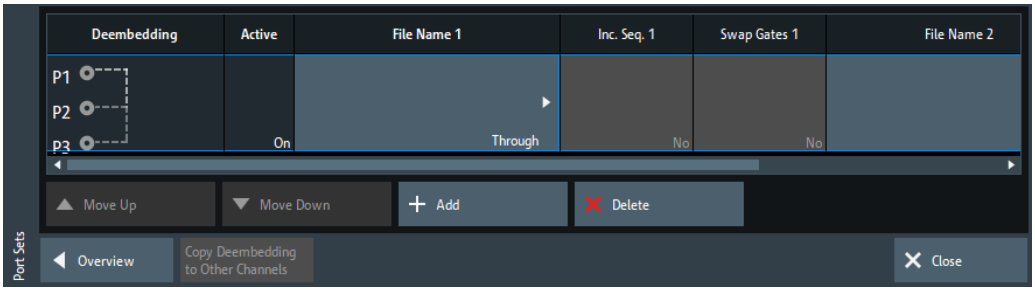
Remote command:
```
CALCulate<Ch>:TRANsform:VNETworks:SENDed:DEEMbedding<PhyPt>:IGNore
```

Copy Deembedding to Other Channels
See "Copy Deembedding to Other Channels" on page 704.

5.13.1.5 Port Sets panel

The "Port Sets" panel allows you to configure the "Port Sets" to whom a deembedding or embedding network can be assigned ("Add", "Delete"). The touchstone files defining these networks can also be selected from here ("...").

Access: Overview panel > "Port Sets"

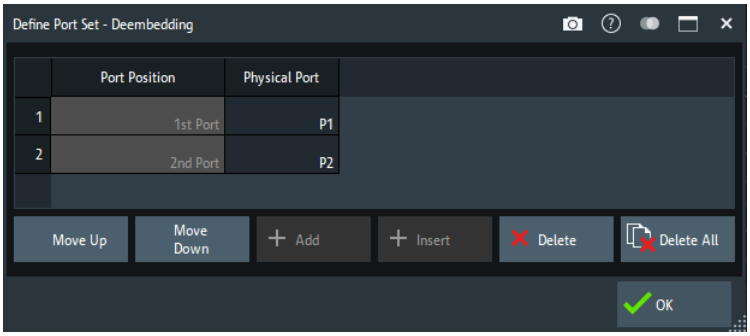


The "Port Sets" panel can also be activated by selecting the Port Sets tab. Refer to its description for background information, parameters and additional remote commands.

Move Up / Move Down
Allows you to modify the sequence in which the active port set deembeddings/embeddings are applied.

Add / Delete
Allows you to define the "Port Sets" to whom a deembedding or embedding network can be assigned.

The "Add" button opens the "Define Port Set" dialog that allows you to define an additional port set, comprising two or more physical ports.



Use the controls on the right-hand side of the dialog to define the port set and the order of the ports within the port set.

Note however that the speed of the de-/embedding calculation depends on the port order. The best perfomance is achieved if the ports are ordered according to their numbers (i.e. in natural order). This is particularly significant for large port sets.

Remote command:

```
CALCulate<Ch>:TRANsform:VNETworks:PPAir:DEEMbedding<ListId>:
DEFine
CALCulate<Ch>:TRANsform:VNETworks:PSET:DEEMbedding<ListId>:
DEFine
CALCulate<Ch>:TRANsform:VNETworks:PPAir:DEEMbedding:DELete
CALCulate<Ch>:TRANsform:VNETworks:PPAir:EMBedding<ListId>:DEFine
CALCulate<Ch>:TRANsform:VNETworks:PSET:EMBedding<ListId>:DEFine
CALCulate<Ch>:TRANsform:VNETworks:PPAir:EMBedding<ListId>:DELete
```

Active
The controls in the "Active" column enable or disable the configured de-/embeding for the related port set (i.e. adds or removes it to/from the calculation flow) without changing its parameters.

Remote command:
```
CALCulate<Ch>:TRANsform:VNETworks:PPAir:DEEMbedding<ListId>[:STATe]
CALCulate<Ch>:TRANsform:VNETworks:PPAir:EMBedding<ListId>[:STATe]
```

File Name <i>/Inc. Seq. <i>/Swap Gates <i>
If, for the respective port set, you select a Balanced tab comprising Touchstone files, then at least one of the related "File Name <i>" cells becomes enabled.

A click or tap opens the Open de-/embedding file dialog.

Note: The loaded file is stored in the active recall set. Recall sets contain the full de-/embedding data so that they can be transferred to other instruments.

For "Inc. Seq. <i>" and "Swap Gates", see "Increasing Port Sequence/Swap Gates" on page 759.

Remote command:
```
MMEMory:LOAD:VNETworks<Ch>:PPAir:DEEMbedding<ListId>
MMEMory:LOAD:VNETworks<Ch>:PPAir:EMBedding<ListId>
```

Copy Deembedding to Other Channels
See "Copy Deembedding to Other Channels" on page 704.

5.13.1.6 Balanced panel

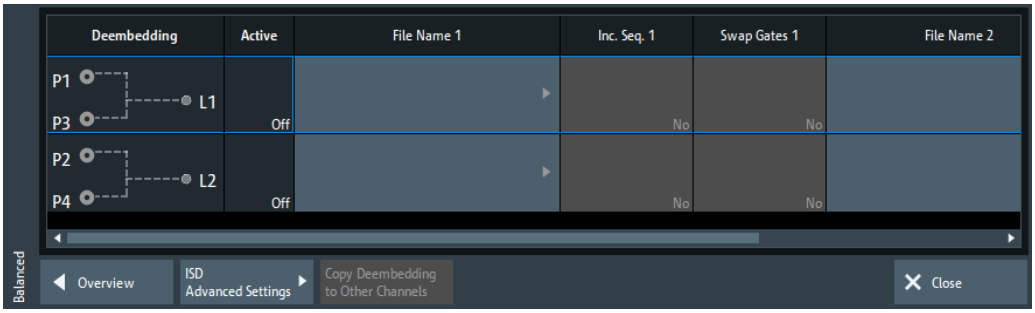
This panel allows you to activate or deactivate deembedding/embedding of balanced ports.

Access: Overview panel > "Balanced"

If the selected Balanced tab is defined using one or more touchstone files, these files can be selected from the dock widget ("...").

For network types that are defined using one or more touchstone files, the required files can also be selected from here ("...").

If necessary, use the button on the right-hand side to open the Balanced Ports dialog and change the balanced port configuration.



This panel can also be activated by selecting the Balanced tab. Refer to its description for background information, parameters and additional remote commands.

Active
The controls in the "Active" column enable or disable the configured de-/embeding for the related balanced port (i.e. adds or removes it to/from the calculation flow) without changing its parameters.

Remote command:
```
CALCulate<Ch>:TRANsform:VNETworks:BALanced:DEEMbedding<LogPt>[:STATe]
CALCulate<Ch>:TRANsform:VNETworks:BALanced:EMBedding<LogPt>[:STATe]
```

File Name <i>/Inc. Seq. <i>/Swap Gates <i>
If, for the respective balanced port, you select a Balanced tab comprising Touchstone files, then at least one of the related "File Name <i>" cells becomes enabled.

A click or tap opens the Open de-/embedding file dialog.

Note: The loaded file is stored in the active recall set. Recall sets contain the full de-/embedding data so that they can be transferred to other instruments.

For "Inc. Seq. <i>" and "Swap Gates", see "Increasing Port Sequence/Swap Gates" on page 759.

Remote command:
```
MMEMory:LOAD:VNETworks<Ch>:BALanced:DEEMbedding<LogPt>
MMEMory:LOAD:VNETworks<Ch>:BALanced:EMBedding<LogPt>
```

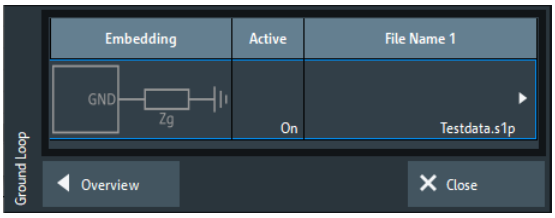
Copy Deembedding to Other Channels
See "Copy Deembedding to Other Channels" on page 704.

5.13.1.7 Ground Loop panel

Allows you to activate or deactivate ground loop deembedding/embedding ("Active").

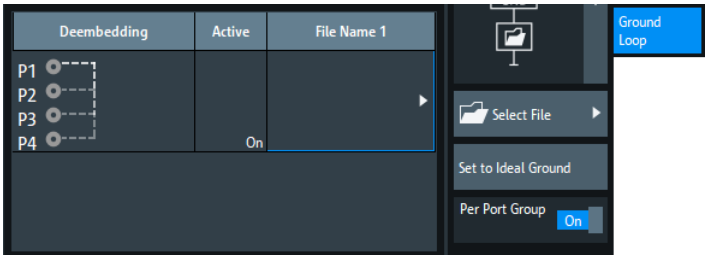
Access: Overview panel > "Ground Loop"

For the "1-Port Data" network type, the required touchstone file can also be selected from here ("...").



This panel can also be activated by selecting the Ground Loop tab softtool tab. Refer to its description for background information, parameters and additional remote commands.

If multiple port groups (DUTs) are configured and Ground Loop per Port Group is enabled, the panel displays the available port groups. Select the port group whose ground loop network shall be configured.



Active
The controls in the "Active" column enable or disable the configured ground loop deembedding/embedding (i.e. adds or removes it to/from the calculation flow) without changing its parameters.

Remote command:
`CALCulate<Ch>:TRANsform:VNETworks:GLOop:DEEMbedding<group>[:STATe]`
`CALCulate<Ch>:TRANsform:VNETworks:GLOop:EMBedding<group>[:STATe]`

File Name 1
If you select the 1-port data Network (`*.s1p` file), then the "File Name 1" cell becomes enabled.

A click or tap opens the Open de-/embedding file dialog.

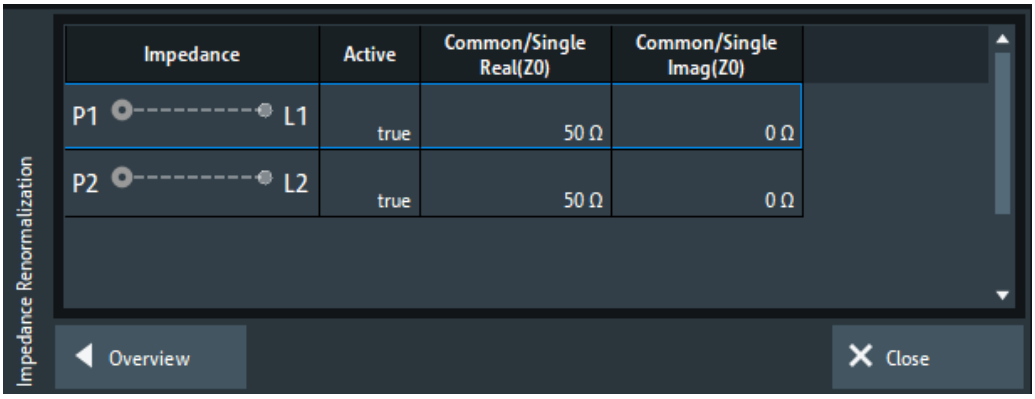
Note: The loaded file is stored in the active recall set. Recall sets contain the full de-/embedding data so that they can be transferred to other instruments.

Remote command:
`MMEMory:LOAD:VNETworks<Ch>:GLOop:DEEMbedding<group>`
`MMEMory:LOAD:VNETworks<Ch>:GLOop:EMBedding<group>`

5.13.1.8 Impedance Renormalization panel

This panel provides alternative access to the reference impedance settings (see "Reference Impedance tab" on page 300).

Access: Overview panel > "Impedance Renor."



Active
The "Active" flags are inversely related to the Use Default flags of the logical port configuration (see Balanced Ports dialog).

"Active"	"Use Default"
☑	☐
☐	☑

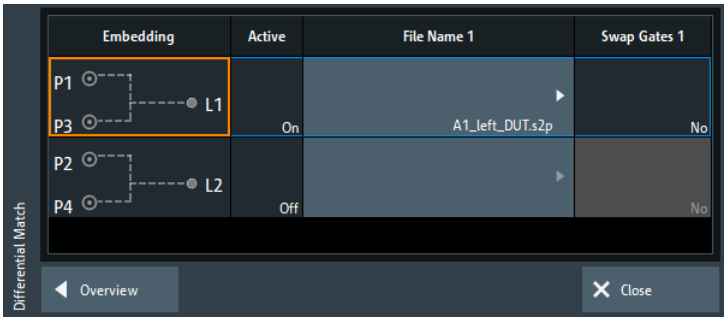
5.13.1.9 Differential Match panel

This panel allows you to activate or deactivate differential match embedding.

Access: Overview panel > "Differential Match"

For the "2-Port Data" network type, the required touchstone file can also be selected from here ("...").

If necessary, use the button on the right hand side to open the Balanced Ports dialog and change the balanced port configuration.



This panel can also be activated by selecting the Differential Match tab softtool tab. Refer to its description of this softtool tab for background information, parameters and additional remote commands.

Active
The controls in the "Active" column enable or disable the configured differential match embedding (i.e. adds or removes it to/from the calculation flow) without changing its parameters.

Remote command:
`CALCulate<Ch>:TRANsform:VNETworks:DIFFerential:EMBedding<LogPt>[:STATe]`

File Name 1/Swap Gates
If you select the "2-Port Data" Network for the respective balanced port, then the corresponding "File Name 1" cell becomes enabled.

A click or tap opens the Open de-/embedding file dialog.

Note: The loaded file is stored in the active recall set. Recall sets contain the full de-/embedding data so that they can be transferred to other instruments.

For "Swap Gates", see "Increasing Port Sequence/Swap Gates" on page 759.

Remote command:
`MMEMory:LOAD:VNETworks<Ch>:DIFFerential:EMBedding<LogPt>`

5.13.1.10 Fixture modeling tool panel

Allows you to model a test fixture using the selected fixture modeling tool and to deembed selected ports using the generated touchstone files.

Supported fixture modeling tools
The following fixture modeling tools are supported:

- EZD
- ISD
- SFD

Access: Channel – [Offset Embed] > "Single Ended" / "Balanced" > "Run Tool"

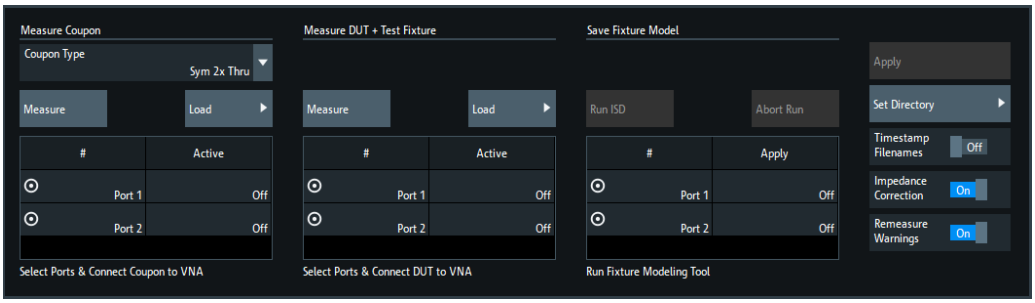


Figure 5-75: Fixture modeling dock widget: single-ended (ISD/SFD/EZD)

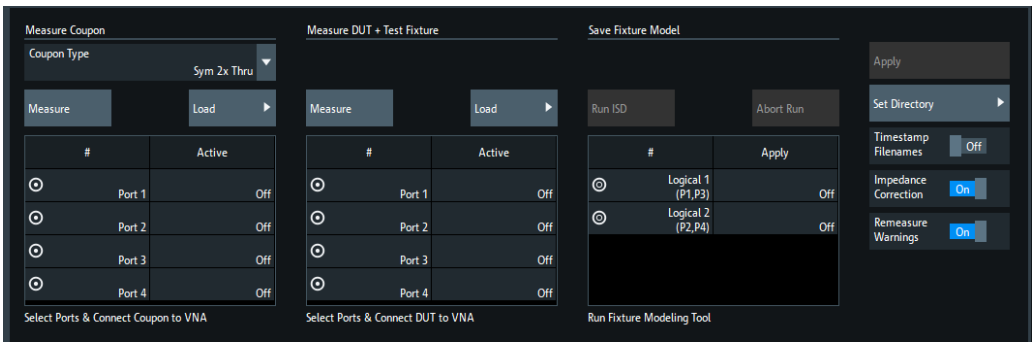


Figure 5-76: Fixture modeling dock widget: balanced (ISD/SFD/EZD)

For ISD, SFD and EZD, the fixture modeling proceeds in the following steps:

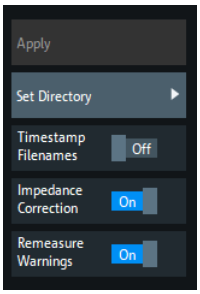
1. Measure one or more test coupons for the related fixture; see "Measure Coupon" on page 717
   The tools differ in the available coupon types.
2. If Impedance Correction shall be performed, measure the DUT with the fixture (see "Measure DUT + Test Fixture" on page 718) or "Load" data from Touchstone file. For EZD, "Load" is disabled if you want to select the "Deembedding Plane" manually (see EZD Advanced Settings).
3. Run the selected tool to calculate the Touchstone files modeling the test fixture (see "Save Fixture Model" on page 719).

   The calculated `sNp` files can then be used to deembed the DUT at selected ports.

- The current implementation assumes symmetrical lead-ins and lead-outs.

Output Settings

This section provides general settings and is located on the right of the ISD/SFD/EZD dock widget.



Set Directory/Timestamp Filenames
Allows you to specify the directory where the fixture modeling tools store their data.

Default is `C:\Users\Public\Documents\Rohde-Schwarz\Vna\Embedding`.

If you activate "Timestamp Filenames", the names of subsequently generated "Test Coupon" and "DUT + Test Fixture" files are prefixed with the current date and time.

Remote command:
CALCulate:FMODel:DIRectory
CALCulate:FMODel:DIRectory:DEFault
CALCulate:FMODel:DIRectory:DEFault:CLEar
CALCulate:FMODel:REName

Impedance Correction
Enables/disables the impedance correction functionality of the respective tool.

This is a global setting.

Remote command:
CALCulate:FMODel:EZD:IMPedance
CALCulate:FMODel:ISD:IMPedance
CALCulate:FMODel:SFD:IMPedance
(CALCulate:FMODel:SFD:AUTO)

Suppress Remeasure Warnings
If enabled, Measure Coupon and Measure DUT + Test Fixture do not raise warnings if the measurement is repeated with different ports. This is a global setting.

Remote command:
n.a.

Advanced Settings ...
Opens the "Advanced Settings" dialog of the selected tool. See:
- "ISD Advanced Settings" on page 720
- "SFD Advanced Settings" on page 724
- "EZD Advanced Settings" on page 725

Reset to Default
Restores the default settings of the selected fixture modeling tool. This comprises:
- The states and settings in the Measure Coupon, Measure DUT + Test Fixture and Save Fixture Model sections (common for all tools)
- The global settings of the selected tool (tool specific)

Remote command:
CALCulate:FMODel:ISD:PRESet
CALCulate:FMODel:SFD:PRESet
CALCulate:FMODel:EZD:PRESet

Apply
This button is only active if the selected deembedding tool has been successfully run (see "Save Fixture Model" on page 719).

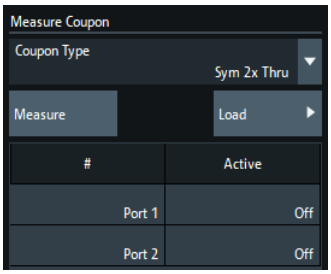
Applies the calculated deembedding files <...>_left_DUT.sNp and <...>_right_DUT.sNp) to the ports marked "Active" in the Save Fixture Model section.

Remote command:
MMEMory:LOAD:VNETworks<Ch>:SENDed:DEEMbedding<PhyPt>
MMEMory:LOAD:VNETworks<Ch>:BALanced:DEEMbedding<LogPt>

Cancel
Closes the dialog without applying any deembedding files.

Measure Coupon

The "Measure Coupon" section allows you to measure one or more test coupons for the related fixture.



Coupon Type
Selects the coupon type to be measured. The following coupon types are supported:

	ISD	SFD	EZD
"Sym 2x Thru"	✓	✓	✓
"1x Open"	✓	✓	–
"1x Short"	✓	✓	–
"1x Open, 1x Short"	✓	–	–

Remote command:
CALCulate:FMODel:ISD:COUPon:TYPE
CALCulate:FMODel:SFD:COUPon:TYPE

Measure
Starts the coupon measurement at the "Active" on page 718"Active" on page 718 ports.

The display area shows all the S-parameter measurements being made. The resulting Touchstone file is written to C:\Users\Public\Documents\Rohde-Schwarz\Vna\Embedding.

For Coupon Type "1x Open, 1x Short" (ISD only) the measurement proceeds in two steps: one for the "1x Open" and one for the "1x Short" coupon. The label of the "Measure" button changes accordingly.

Remote command:
CALCulate:FMODel:ISD:COUPon:MEASure
CALCulate:FMODel:ISD:COUPon:MEASure:OPEN
CALCulate:FMODel:ISD:COUPon:MEASure:SHORt

CALCulate:FMODel:SFD:COUPon:MEASure
CALCulate:FMODel:EZD:COUPon:MEASure

Load File
Allows you to load the coupon properties from a Touchstone file (*.s*p").

Remote command:
CALCulate:FMODel:ISD:COUPon:MEASure:FILename
CALCulate:FMODel:ISD:COUPon:MEASure:FILename:CLEar
CALCulate:FMODel:ISD:COUPon:MEASure:OPEN:FILename
CALCulate:FMODel:ISD:COUPon:MEASure:SHORt:FILename
CALCulate:FMODel:SFD:COUPon:MEASure:FILename
CALCulate:FMODel:SFD:COUPon:MEASure:FILename:CLEar
CALCulate:FMODel:EZD:COUPon:MEASure:FILename
CALCulate:FMODel:EZD:DUT:MEASure:FILename:CLEar

Active
Before measuring the test coupon, use these controls to indicate the physical ports to which it is connected.

Table 5-7: Allowed numbers of active ports

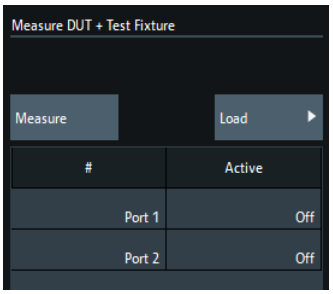
	"Sym 2x Thru"	"1x Open, 1x Short"	other
Single-ended	2	1 or 2	1
Balanced	2 or 4	2 or 4	2

Remote command:
CALCulate:FMODel:ISD<Ph_pt>:COUPon[:STATe]
CALCulate:FMODel:SFD<Ph_pt>:COUPon[:STATe]
CALCulate:FMODel:EZD<Ph_pt>:COUPon[:STATe]

Measure DUT + Test Fixture

The "Measure DUT + Test Fixture" section allows you to measure the whole structure, i.e. the DUT with the test fixture.

This section is only available if impedance correction is enabled.



Load File
Load DUT + test fixture data from file

Remote command:
CALCulate:FMODel:EZD:DUT:MEASure:FILename
CALCulate:FMODel:EZD:DUT:MEASure:FILename:CLEar
CALCulate:FMODel:ISD:DUT:MEASure:FILename
CALCulate:FMODel:ISD:DUT:MEASure:FILename:CLEar
CALCulate:FMODel:SFD:DUT:MEASure:FILename
CALCulate:FMODel:SFD:DUT:MEASure:FILename:CLEar

Measure
Starts the measurement of DUT + test fixture at the Active ports.

The display area shows all the S-parameter measurements being made. The resulting Touchstone file is written to C:\Users\Public\Documents\Rohde-Schwarz\Vna\Embedding.

Remote command:
CALCulate:FMODel:EZD:DUT:MEASure
CALCulate:FMODel:ISD:DUT:MEASure
CALCulate:FMODel:SFD:DUT:MEASure

Active
Before starting to Measure, use these controls to indicate the physical ports to which DUT + test fixture are connected. For single-ended deembedding, 2 ports must be active, for balanced deembedding 4 ports.

Remote command:
CALCulate:FMODel:EZD<Ph_pt>:DUT[:STATe]
CALCulate:FMODel:ISD<Ph_pt>:DUT[:STATe]
CALCulate:FMODel:SFD<Ph_pt>:DUT[:STATe]

Save Fixture Model

This section allows you to run the selected deembedding tool to generate the Touchstone files for fixture deembedding (and the Touchstone file of the de-embedded DUT).

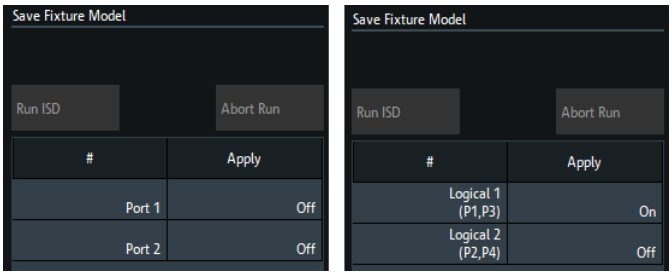


Figure 5-77: Save Fixture Model section

left = single-ended
right = balanced

Run <Fixture Modeling Tool>
Runs the selected fixure deembedding tool.

This button is only active if:

- The test coupon measurement finished successfully or the test coupon data were successfully loaded from file (see "Measure Coupon" on page 717)
- The measurement of DUT + test fixture finished successfully (see "Measure DUT + Test Fixture" on page 718)

The resulting Touchstone files (S-parameters of the lead-ins, the lead-outs, and the de-embedded DUT) are written to C:\Users\Public\Documents\Rohde-Schwarz\Vna\Embedding. If result files with the same names already exist, you can decide whether you want to rename or overwrite them.

If running the deembedding tool causes errors, the <Deembedding Tool> Errors dialog provides additional information.

Note: For ISD, first a batch task file `Embeddingconfig_znb.abt` is created and then the tool is run in batch mode (see the ISD User Guide).

Remote command:

```
CALCulate:FMODel:ISD:RUN:RUN
CALCulate:FMODel:SFD:RUN:RUN
CALCulate:FMODel:EZD:RUN:RUN
```

Apply

Use the controls in the "Apply" section to select the ports to which you want to assign the generated deembedding files when the dialog is closed using the Apply button.

Note: For a "left" deembedding file, the original port order is used, for a "right" deembedding files the swap gates (ports) function is used.

Remote command:

```
CALCulate:FMODel:ISD<Pt>:RUN[:STATe]
CALCulate:FMODel:SFD<Ph_pt>:RUN[:STATe]
CALCulate:FMODel:EZD<Pt>:RUN[:STATe]
```

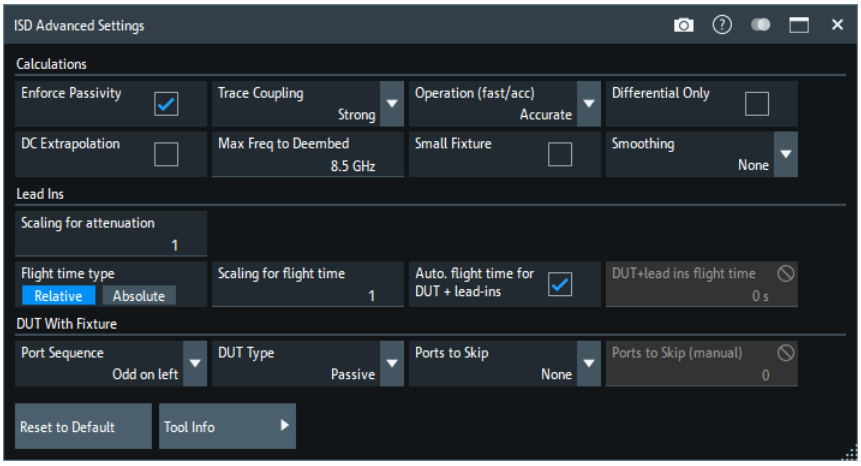
<Deembedding Tool> Errors dialog

If running the related deembedding tool causes errors, a dialog provides additional information:

ISD Advanced Settings

Advanced settings of the ISD tool. For details, see the ISD User Guide.

For the analyzer firmware, these settings are global.



Calculations
Some additional parameters that define how the ISD tool calculates its output.

Enforce Passitivity ← Calculations
If enabled (default), the ISD tool enforces passivity and reciprocity for the test coupons and the test fixture.

Remote command:
`CALCulate:FMODel:ISD:PASSivity`

Trace Coupling ← Calculations
Tells the ISD tool about coupling among lead-in traces

- "None": No coupling.
  The ISD tool optimizes odd- and single-ended insertion losses.
- "Weak": Coupling is extracted even if there are 2 ports enabled on the test coupon
- "Strong" (default): If the test coupon is a 4-port file, and there are two ports to be extracted, the ISD tool optimizes odd- and even-mode insertion losses.
  The ISD tool automatically reverts to "Weak" if these conditions are not met.

Remote command:
`CALCulate:FMODel:ISD:TRACe:COUPling`

Operation (fast/acc) ← Calculations
Influences the speed of execution and accuracy of the ISD tool.

- "Accurate" (default): Normal execution mode
- "Fast": reduces the execution time to ~50%
  Often this mode can be enabled with only little loss of accuracy.

Remote command:
`CALCulate:FMODel:ISD:OPERation`

Differential Only ← Calculations
Use this option to remove common mode interference to improve higher frequency resonance effect.

Remote command:
`CALCulate:FMODel:ISD:DIFFonly`

Max Freq to Deembed ← Calculations
When the test coupon's return loss is not less than the insertion loss, the maximum frequency to deembed together with the proper Insertion Loss setting gives the best accuracy.

By default, the maximum frequency is set to the maximum frequency the VNA can use.

Remote command:
`CALCulate:FMODel:ISD:SCALe:FREQuency`

DC Extrapolation ← Calculations
This setting is only active if ISD version 2019.12.18 or later is used.

If disabled (default), the touchstone data of the generated deembedding file reach down to the start frequency of the instrument sweep. If enabled, the ISD tool adds (extrapolated) S-parameter data for f = 0.

Remote command:
`CALCulate:FMODel:ISD:DCEXtrapolat`

Small Fixture ← Calculations
Enables/disables the ISD tool's small fixture mode.

Remote command:
`CALCulate:FMODel:ISD:SMALlfixture`

Smoothing ← Calculations
This setting is only visible if ISD version 2023.05.15 or later is used.

Select the appropriate smoothing level ("None", "Smooth," "Smoother", "Smoothest" and "Smoothest + DUT's IL and RL") to smooth out insertion and return loss ripples of the DUT. Best used for reciprocal and passive structures with insertion loss. "Smoothest" is recommended for non-resonant DUTs, "Smooth" or "None" for resonant DUTs. For detailed information, see the ISD user guide.

Remote command:
`CALCulate:FMODel:ISD:SMOThing`

Test Coupons > Insertion Loss
Tells the ISD tool about the linearity of the 2xThru test coupon:
- "Linear": Linear insertion loss
- "Non-Linear" (default): Nonlinear insertion loss
- "Resonant": The 2x Thru test coupon is split and used directly for deembedding This option can be more accurate if the fixture and the 2x Thru have the same impedance at every location.

Remote command:
`CALCulate:FMODel:ISD:ATTenuation:BEHavior`

Lead Ins
Defines how the ISD tool interprets the lead-in data.

Scaling for attenuation ← Lead Ins
Scales the test coupon's attenuation.

Default is 1.

Remote command:
CALCulate:FMODel:ISD:SCALe:ATTenuation

Flight Time Type/Scaling for flight time ← Lead Ins
"Scaling for flight time" overrides the lead-in's flight time (i.e. its delay) in case the through-trace test coupon is a bit too short or too long.

You can specify the value in relative and in absolute terms ("Flight Time Type"). "Relative" and 1 is default (no scaling).

Remote command:
CALCulate:FMODel:ISD:SCALe:FTIMe:TYPE
CALCulate:FMODel:ISD:SCALe:FTIMe

Auto. flight time for DUT + lead-Ins ← Lead Ins
If enabled (default) the flight time for DUT + lead-ins is calculated automatically. Otherwise it can be specified manually (see "DUT + lead-ins flight time" on page 723).

Remote command:
CALCulate:FMODel:ISD:FTIMe:OVERride

DUT + lead-ins flight time ← Lead Ins
Allows you to set the flight time for DUT + lead-ins manually, if Auto. flight time for DUT + lead-Ins is deactivated,

Set the value manually, if the "DUT + Test Fixture" does not have a Through response and the total flight time exceeds 10ns.

Remote command:
CALCulate:FMODel:ISD:FTIMe:DUT

DUT With Fixture
Defines how the ISD tool interprets the "DUT + Test Fixture" data.

Port Sequence ← DUT With Fixture
Tells the ISD tool about the arrangement of "DUT + Fixture" ports:
- "1 to N on left": Ports 1 to N are on the left and ports N+1 to 2*N are on the right.
- "Odd on left" (default): 1, 3, 5, etc. are on the left and ports 2, 4, 6, etc. are on the right.
- "All on left": All ports are on the left

Note: Ports on the left/right are assumed to be coupled if Trace Coupling is not set to "None". A port on the left side is not considered to be coupled to a port on the right side.

Remote command:
CALCulate:FMODel:ISD:PORT:ORDer

DUT Type ← DUT With Fixture
Select "Passive"/"Active" if the DUT is passive/active.

Default is "Passive".

Remote command:
CALCulate:FMODel:ISD:DUT:TYPE

Ports to Skip ← DUT With Fixture
Tells the ISD tool which ports (in the measured "DUT + Test Fixture" file) to skip when the tool is run.

- "None" (default): do not skip any ports
- "Ports on left": Skip the ports on the left (according to the selected Port Sequence)
- "Ports on right": Skip the ports on the right (according to the selected port sequence)
- "Manually set ports": use Ports to Skip (manual) to define the ports to be skipped

Remote command:
```
CALCulate:FMODel:ISD:PORT:SKIP:NONE
CALCulate:FMODel:ISD:PORT:SKIP:LEFT
CALCulate:FMODel:ISD:PORT:SKIP:RIGHt
```

Ports to Skip (manual) ← DUT With Fixture
If Ports to Skip is set to "Manually set ports", this text field allows you to define the ports to be skipped. Enter the port numbers, separated by blanks (e.g. '1 3 4').

Remote command:
```
CALCulate:FMODel:ISD:PORT:SKIP
```

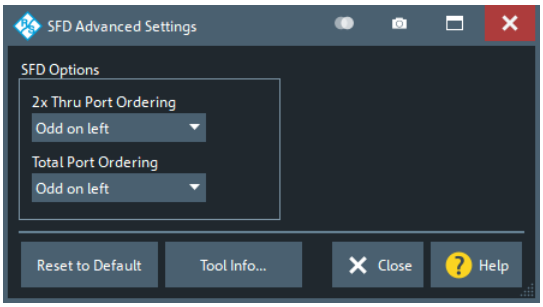
Reset to Default
Resets the settings of the "ISD Advanced Settings" to their defaults.

Remote command:
```
CALCulate:FMODel:ISD:PRESet
```

Tool Info
Displays additional information about the current version of the ISD tool.

SFD Advanced Settings

Advanced settings of the SFD tool. For the analyzer firmware, these settings are global.



2x Thru Port Ordering
Tells the SFD tool about the port ordering of the 2x Thru test coupon

- "Ports 1&3 on Left" (default): Odd ports are on the left and even ports are on the right
- "Ports 1&2 on Left" ports 1 to N are on the left and ports N+1 to 2·N are on the right

Remote command:
CALCulate:FMODel:SFD:DIFFcfg

Total Port Ordering
Tells the SFD tool about the port ordering of the test fixture
- "Ports 1&3 on Left" (default): Odd ports are on the left and even ports are on the right
- "Ports 1&2 on Left" ports 1 to N are on the left and ports N+1 to 2·N are on the right

Remote command:
CALCulate:FMODel:SFD:TOTaldiffcfg

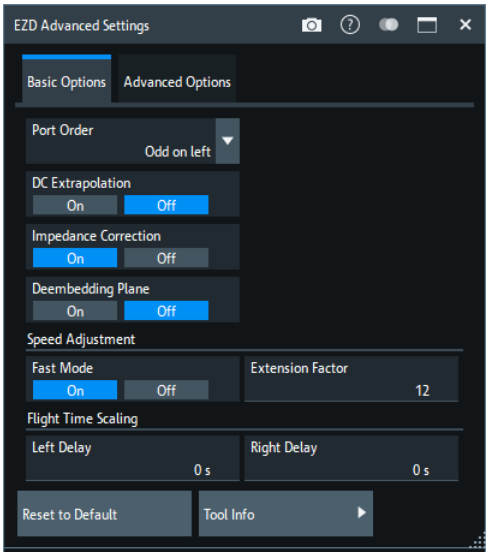
Reset to Default
Resets the settings of the "SFD Advanced Settings" to their defaults.

Remote command:
CALCulate:FMODel:SFD:PRESet

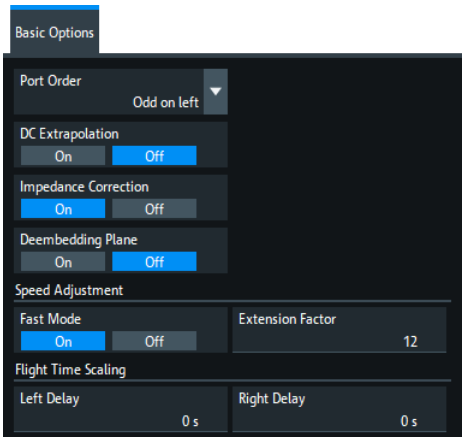
Tool Info
Displays additional information about the current version of the SFD tool.

EZD Advanced Settings

Advanced settings of the EZD tool. For the analyzer firmware, these settings are global.



Basic Options tab



Port Order
Tells the EZD tool about the port ordering of the test coupon.

- "Odd on left" (default): Odd ports are on the left and even ports are on the right.
- "1 to N on left" ports 1 to N are on the left and ports N+1 to 2·N are on the right.

Remote command:
`CALCulate:FMODel:EZD:PORT:ORDer`

DC Extrapolation
If disabled (default), the touchstone data of the generated deembedding file reach down to the start frequency of the instrument sweep. If enabled, the EZD tool adds (extrapolated) S-parameter data for f = 0.

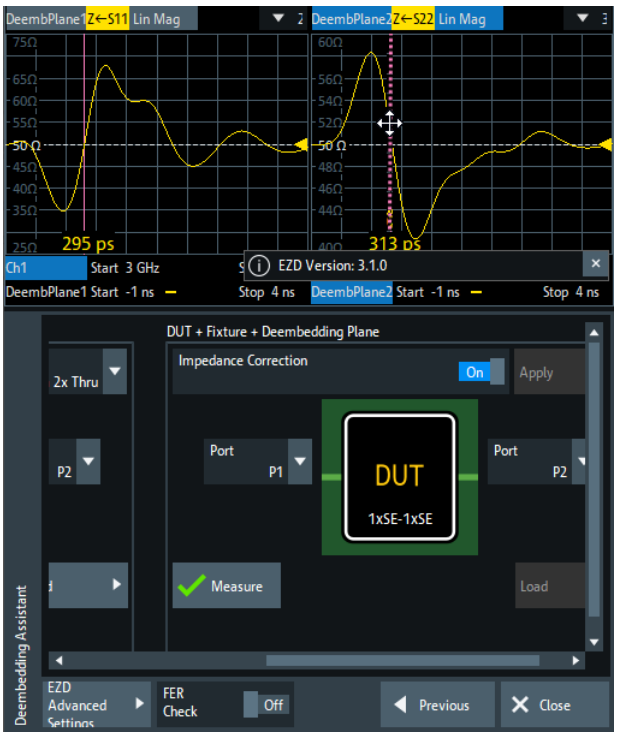
Remote command:
`CALCulate:FMODel:EZD:DCEXtrapolat`

Impedance Correction
Activates impedance correction (i.e. the "DUT + Test Fixure") step in the fixture deembedding dock widgets.

Remote command:
`CALCulate:FMODel:EZD:IMPedance`

Deembedding Plane
This switch is only enabled, if "Impedance Correction" on page 726 is active.

If active, then while you *measure* "DUT + Fixure", you can select the position of the deembedding plane. Simply drag the purple vertical lines on the left ("DeembPlane1") and on the right ("DeembPlane2") to the appropriate position in the TDR impedance profile traces.



This input sets the pullback of the EZD algorithm.

Note:

- To facilitate iterations, the "Deembedding Plane" are kept as global values.
- Flight Time Scaling is ignored if "Deembedding Plane" is active.

Remote command:

`CALCulate:FMODel:EZD:DEPLane:STATe`

`CALCulate:FMODel:EZD:DEPLane<Pl>`

Speed Adjustment

"Fast Mode" reduces runtime by resampling the calibration and "DUT + Fixture"" to a coarser frequency grid, and by using a faster DC extrapolation than the iterative method defined in IEEE 370. Results are typically very close to the "non-fast" mode, with possibly some degradation at the lowest frequencies. Deactivate "Fast Mode" when you need maximum accuracy, or are troubleshooting low frequency behavior.

The "Extension Factor" is an advanced setting intended for users who need to tune the fast-mode grid or Impedance Correction. Controls the frequency grid density when "Fast Mode" is on. Higher values give more points (slower, often more accurate); lower values give fewer points (faster). Default is 12; suggested range is 4–20.

Remote command:

`CALCulate:FMODel:EZD<Ph_pt>:FASTmode`

Flight Time Scaling

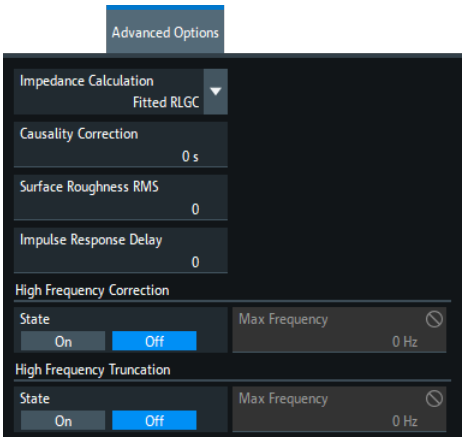
Flight time scaling is used to change the location in the coupon/DUT time domain response where EZD assumes the actual physical device begins (in units of seconds from t=0). It can be useful for removing reflections from probe tips contacting the device measurement pads or other time domain artifacts that should not contribute to the deembedding result.

"Flight Time Scaling" is ignored if Deembedding Plane is active.

Remote command:

```
CALCulate:FMODel:EZD<Ph_pt>:DELay:LEFT
CALCulate:FMODel:EZD<Ph_pt>:DELay:RIGHt
```

Advanced Options tab



Impedance Calculation

Controls how the fixture's characteristic impedance is derived during error-box extraction. The choice trades off accuracy (e.g. frequency-dependent or fitted impedance) with simplicity and compatibility with older methods.

Fitted RLGC	This technique uses a least squares fit of the attenuation vs. frequency relationship to establish a nominal attenuation curve for computing loss over distance for the deembedding. It is slower but can be more robust to measurement noise and variance.
Direct RLGC	This technique uses the raw measurements data point by point to compute the attenuation vs. frequency relationship. There is no attempt to smooth or fit the data.
Constant	This technique assumes a constant impedance over the length of the coupon.

Remote command:

```
CALCulate:FMODel:EZD<Ph_pt>:ICALculation
```

Causality Correction

Causality correction compensates for non-causal energy before t=0 in the "DUT + Fixture" response – often from calibration artifacts or measurement limitations.

If the coupon measurements show a response before t=0, setting "Causality Correction" to a value large enough to envelop that response will effectively ignore it.

Remote command:
CALCulate:FMODel:EZD<Ph_pt>:CAUSality

Surface Roughness RMS
If the Impedance Calculation method is set to "Fitted RLGC", "Surface Roughness RMS" is the conductor surface roughness (RMS height) used. It improves the fitting of attenuation and impedance by accounting for the effect of roughness on conductor loss. It is used to compute a frequency-dependent roughness factor that scales the conductor loss in the fit, giving a more accurate attenuation and impedance for rough conductors.

Remote command:
CALCulate:FMODel:EZD<Ph_pt>:SURFace

Impulse Response Delay
Tweak this value to align the reference plane with a specific electrical or physical location or to correct calibration delay. The default 0 is appropriate for most cases.

Remote command:
n.a.

High Frequency Correction
When "State" is OFF, all S-parameters are used in the deembedding process. When ON, a fitting function is used above the "Max. Frequency" to smooth the S-parameters, and potentially improve the deembedding results for the high frequency content.

If enabled, "Max. Frequency" determines the frequency point above which a fitting function for the S-parameters is used.

Remote command:
CALCulate:FMODel:EZD<Ph_pt>:HFC
CALCulate:FMODel:EZD:IMPedance:BWIDlimit

High Frequency Truncation
Limits the input frequency range: all loaded S-parameter data are cropped from DC (or the existing start) up to the specified "Maximum Frequency" before any deembedding runs. Use this function to focus on a band of interest or to reduce data size and runtime.

If enabled, the calibration network(s) and the fixture DUT fixture (total structure) are cropped so that no frequency point exceeds the given "Maximum Frequency". Only the range from the lowest frequency up to this maximum is used. This allows you to restrict the analysis to a band of interest, reduce the amount of data processed, or shorten runtime.

Remote command:
n.a.

Common controls

Reset to Default
Resets the settings of the "EZD Advanced Settings" to their defaults.

Remote command:
CALCulate:FMODel:EZD:PRESet

Tool Info
Displays additional information about the current version of the EZD tool.

Tool Info dialog

Displays additional information about the related fixture deembedding tool.

Access:

- ISD Advanced Settings > "Tool Info"
- SFD Advanced Settings > "Tool Info"
- EZD Advanced Settings> "Tool Info"

Version
The currently installed version of the related fixture deembedding tool.

Remote command:
CALCulate:FMODel:EZD:VERSion?
CALCulate:FMODel:ISD:VERSion?
CALCulate:FMODel:SFD:VERSion?

5.13.1.11 Deembedding Assistant panel

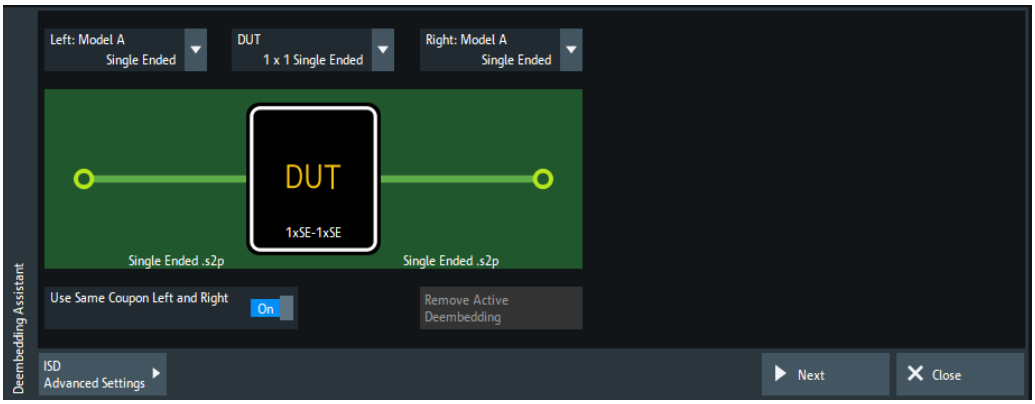
The two pages of the "Offset Embed" dock widget's "Deembedding Assistant" panel allow you to model your DUT and its test fixture, to collect the input data for the selected fixture tool, to run the tool, and to apply the calculated deembedding.

Access: Channel – [Offset Embed] > "Deembed Assistant" tab

(Click the tab to reopen)

Page 1: Topology

The first page of the "Deembedding Assistant" defines the deembedding topology. It allows you to describe your device under test (DUT) in terms of inputs, outputs, and the test coupons required to model the DUT's fixture.



Depending on the selected fixture tool, it narrows down the selectable DUT types and test coupon models.

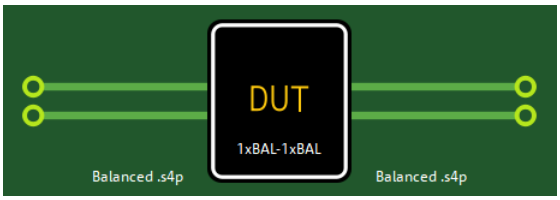
For DeltaCal deembedding, only single-ended DUTs and fixture models are supported.

DUT
The first step in setting up a new deembedding with the "Deembedding Assistant" is to select the type of DUT. Depending on the number of available test/DUT ports and the selected fixture tool, choose one of the following DUT types:

Table 5-8: DUT types

DUT type	Description	ISD only
"1 x 1 Single Ended"	1 single-ended input, 1 single-ended output	☐
"1 x Single Ended"	1 single-ended input, no output	☐
"1 x 1 Balanced"	1 balanced input, 1 balanced output	☐
"1 x Balanced"	1 balanced input, no output	☐
"1 x Balanced, 1 x Single Ended"	1 balanced input, 1 single-ended output	☑
"2 x 1 Single Ended"	2 single-ended inputs, 1 single-ended output	☑
"2 x 2 Single Ended"	2 single-ended inputs, 2 single-ended outputs	☑
"n x Single Ended"	n single-ended inputs, no outputs	☑
"n x m Single Ended"	n single-ended inputs, m single-ended outputs	☑
"n x Balanced"	n balanced inputs, no outputs	☑
"n x m Balanced"	n balanced inputs, m balanced outputs	☑
"n x n Multiport"		☑

The dock widget presents a graphical representation of the selected "DUT" and coupon models, e.g. for a "1 x 1 Balanced" DUT witch "Balanced" left and right models:



Remote command:
`CALCulate:FMODel:DEASsistant:DUT:TYPE`
`CALCulate:FMODel:DEASsistant:PRESet`

Left: Model A / Right: Model A|B
Allows you to select a coupon type, i.e. to define how the coupling at the left side (lead-in) and right side (lead-out) of the DUT is modeled.

If you select the same model for the left and right side and enable "Use Same Coupon Left and Right" on page 733, then also the right side is labeled with "Model A" and in the measurement phase only the data of a single coupon ("Coupon A") must be measured or loaded from file.

"Single Ended" This coupon model is appropriate for one or more single-ended inputs (outputs), where:

- All inputs (outputs) use the same coupon model and
- There is no interaction between inputs (outputs) if more than one is present on the DUT

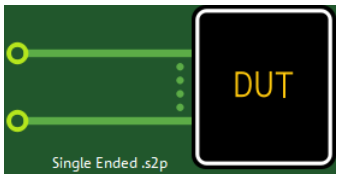


Figure 5-78: n single-ended input ports (uncoupled)

"Balanced" This coupon type is appropriate for one or more balanced inputs (outputs), if:

- All inputs (outputs) use the same coupon model and
- There is no interaction between inputs (outputs) if more than one is present on the DUT

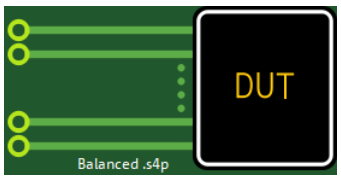


Figure 5-79: n balanced input ports

"Uncoupled" This coupon type selection is available only for balanced ports on the DUT. It is used for deembedding a balanced port when it is desired to treat the single-ended legs of the port as uncoupled entities.

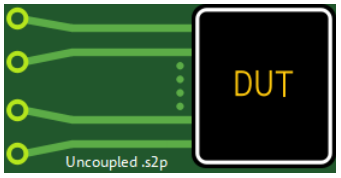


Figure 5-80: n uncoupled balanced input ports

"Coupled" This coupon type is available for DUTs with two or more single-ended or balanced inputs (outputs). It is suitable for a multiport DUT with coupled inputs (outputs) that have to be deembedded as a single unit.
Coupled coupons are drawn with a brace to highlight the difference between coupled and uncoupled ports.

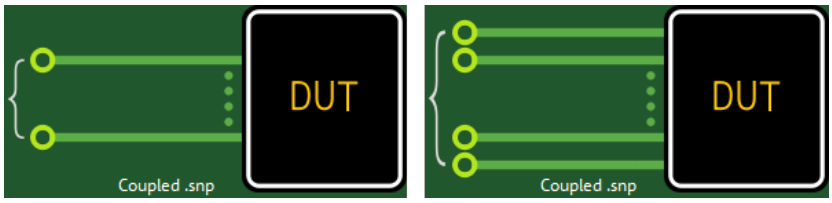


Figure 5-81: n > 1 coupled input ports (single-ended | balanced)

Remote command:
`CALCulate:FMODel:DEASsistant:FIXTure:LEFT:TYPE`
`CALCulate:FMODel:DEASsistant:FIXTure:RIGHt:TYPE`
`CALCulate:FMODel:DEASsistant:PRESet`

Use Same Coupon Left and Right
This control is only active if you selected a two-sided DUT and the same coupon model for the left and right side.

If enabled, then in the measurement phase only the data of a single coupon ("Coupon A") must be measured or loaded from file.

Note: For DeltaCal this is always true and hence this control is hidden.

Remote command:
`CALCulate:FMODel:DEASsistant:SAMCoupon`
`CALCulate:FMODel:DEASsistant:PRESet`

Remove Active Deembedding
Use this button to remove the fixture deembedding and to deactivate all other deembedding in the active channel (see"All Deembedding Activated" on page 747).

Note that the "Deembedding Assistant" works slightly different than the legacy Fixture modeling tool panel. In the "Deembedding Assistant", once applied, the deembedding remains active until cleared or (partly) overwritten. The "Fixture Modeling" dialog resets all deembedding files of a specific type (single-ended, balanced, port set), every time you press the "Apply" button.

Overview
This button takes you to the Overview panel of the "Offset Embed" dock widget, which allows you to interact with the individual components of the deembedding calculation chain.

To reopen the "Deembedding Assistant" panel, you have to click/tap on the "Deembed Assistant" tab caption once again.

<Tool> Advanced Settings
Opens the "Advanced Settings" dialog of the selected fixture tool:
- "ISD Advanced Settings" on page 720

- "SFD Advanced Settings" on page 724
- "EZD Advanced Settings" on page 725

Page 2: Measure / Load and Apply

The second page of the "Deembedding Assistant" dock widget allows you to:

- Measure one or more test coupons, or load their S-parameter data from file
- Measure DUT + test fixture
- Send the measured/loaded data to the selected fixture tool and apply the calculated deembedding



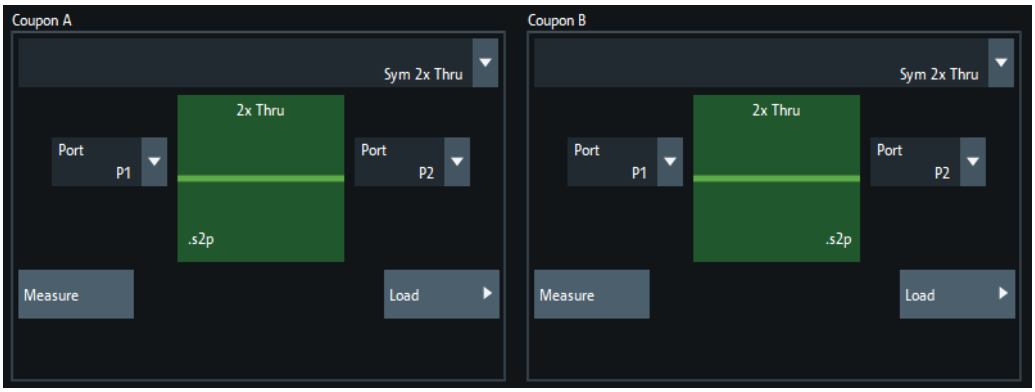
Figure 5-82: Measure / Load and Apply for ISD (SFD, EZD similar)

If you select Use Same Coupon Left and Right on the topology page, then only one set of test coupons can be measured (or loaded from file). The results are used for both sides of the DUT.

With two coupons, "Coupon A" is used to deembed the left-hand side of the DUT and "Coupon B" is used to deembed the right-hand side.

Coupon A / Coupon B section

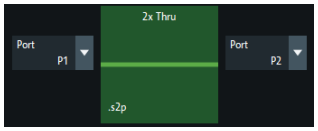
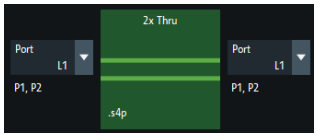
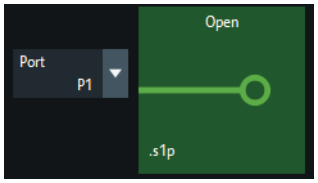
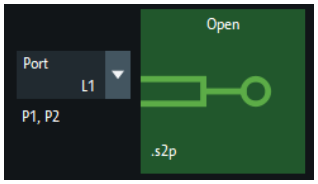
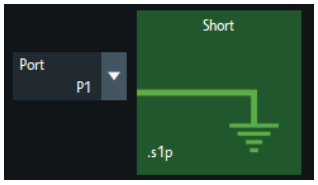
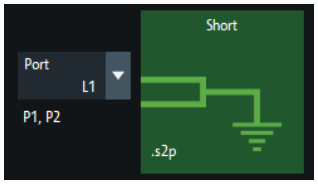
In the "Coupon A" and "Coupon B" section, you can select one or more test coupons and, for each of them, either measure its S-parameters at selected logical ports or load them from a Touchstone file.



Depending on the selected DUT, a different set of test coupon types is supported.

Test coupon	"Sym 2x Thru"	"1x Open"	"1x Short"	"1x Open, 1x Short"
ISD	☑	☑	☑	☑
SFD	☑	☑	☑	☐
EZD	☑	☐	☐	☐

Depending on the selected test coupon type, the number of logical ports and the graphical representation changes:

Test coupon type	Single-ended	Balanced
"Sym 2x Thru"		
"1x Open"		
"1x Short"		

Use the Balanced Ports dialog to create the single-ended and balanced ports that are required for the measurements.

There are several rules to be considered with the coupon measurement interface.

- The port selection drop-down allows you to select from any available port of the correct type on the instrument.
  - For single-ended coupons, any physical port can be selected, even if that port is part of a logical balanced port or port set.
  - For balanced coupons, you can only select a preciously defined logical port.
- The ports used for the measurement must be unique. That is, you cannot select P1 for both sides of a "Sym 2x Thru" coupon. However, you can select the same port for both measurements in a "1x Open, 1x Short" coupon, which requires a connection change before the second measurement.
- The "Load" button allows you to load a Touchstone file instead of measuring the coupon. The graphic displays the Touchstone file's required "cardinality".

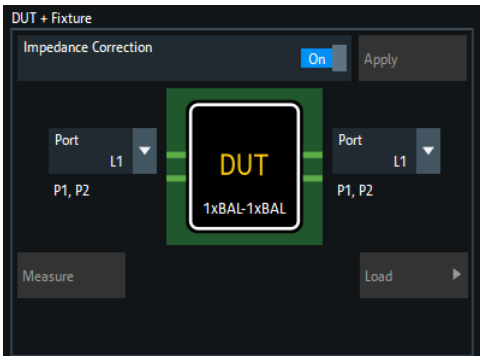
Remote command:

```
CALCulate:FMODel:DEASsistant:COUPon:LEFT:TYPE
CALCulate:FMODel:DEASsistant:COUPon:RIGHt:TYPE
CALCulate:FMODel:DEASsistant:COUPon:LEFT:PORT
CALCulate:FMODel:DEASsistant:COUPon:RIGHt:PORT
```

```
CALCulate:FMODel:DEASsistant:COUPon:LEFT:MEASure
CALCulate:FMODel:DEASsistant:COUPon:RIGHt:MEASure
CALCulate:FMODel:DEASsistant:COUPon:LEFT:FILE
CALCulate:FMODel:DEASsistant:COUPon:RIGHt:FILE
CALCulate:FMODel:DEASsistant:COUPon:LEFT:CLEar
CALCulate:FMODel:DEASsistant:COUPon:RIGHt:CLEar
```

DUT + Fixture section

After you have measured the coupons or loaded their data from file, you can proceed with the DUT and test fixture assembly. Either measure the assembly, or load its data from file.

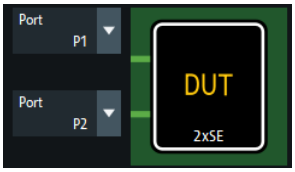


For measuring coupons, choose:

- A single-ended or balanced port for the 1x DUT sides



- Two single-ended ports for the 2x single-ended DUT sides



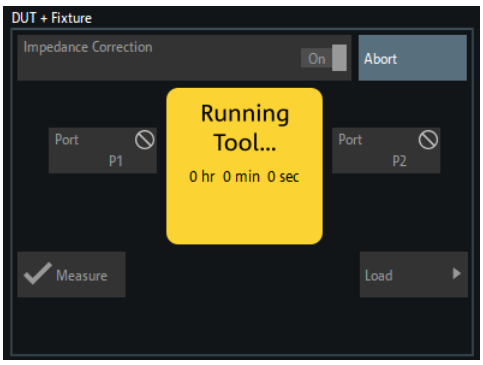
- A range of single-ended or balanced ports for the nx DUT sides



If you want to apply impedance correction, switch "Impedance Correction" ON and "Measure" the DUT with the fixture or "Load" the data from Touchstone file.

For EZD, "Load" is disabled if you want fo select the "Deembedding Plane" manually (see EZD Advanced Settings).

Once the coupons (and the DUT) are measured, the "Apply" button becomes active. Use it to run the tool and, if successful, perform the deembedding on the selected ports.



The deembedding of single-ended and decoupled ports is assigned as Single Ended, the deembedding of balanced ports as Balanced, and the deembedding of coupled as Port Sets deembedding.

Remote command:

```
CALCulate:FMODel:DEASsistant:IMPCorrect
CALCulate:FMODel:DEASsistant:DUT:PORT
CALCulate:FMODel:DEASsistant:DUT:MEASure
CALCulate:FMODel:DEASsistant:RUN
CALCulate:FMODel:DEASsistant:DUT:FILE
CALCulate:FMODel:DEASsistant:DUT:CLEar
```

5.13.2 Offset tab

Defines length offset parameters for each port.

Use the complementary dock widget to activate or deactivate length/loss compensation for selected ports (see Section 5.13.1.3, "Offset panel", on page 705).

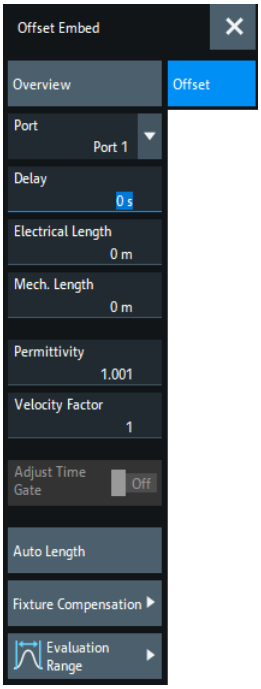
The marker function Zero Delay at Marker function overwrites the offset parameters.

Background information

Refer to the following sections.

- Section 4.6, "Offset parameters and de-/embedding", on page 197
- Section 4.6.1.1, "Definition of offset parameters", on page 198
- Section 4.6.1.3, "Auto Length", on page 199
- Section 4.6.1.6, "Application and effect of offset parameters", on page 202
- Section 4.6.1.7, "Offset parameters for balanced ports", on page 203
- Section 4.6.1.5, "Fixture Compensation", on page 201

5.13.2.1 Controls on the Offset tab



Overview
This button is available on all "Offset Embed" softtool tabs. It opens the Overview panel in the Offset Embed dock widget.

Port
Physical test port of the analyzer. You can define independent offset parameters for all ports.

Remote command:
The `<PhyPt>` numeric suffix in the `[SENSe<Ch>:]CORRection:...` commands identifies the physical port.

Delay / Electrical Length / Mech. Length
Defines the length offset at the selected port as a delay, an electrical length, or a mechanical length. The three quantities are related by:

$$Delay = \frac{L_{mech} \cdot \sqrt{\varepsilon_r}}{c};\ Electrical\ Length = L_{mech} \cdot \sqrt{\varepsilon_r}$$

and overwrite each other. See also Section 4.6.1.1, "Definition of offset parameters", on page 198.

Note: The entered parameters must correspond the actual (one-way) length of the transmission line. To account for the propagation in both directions, the phase shift of a reflection parameter due to a given length offset is twice the phase shift of a transmission parameter. For a numeric example, see Section 4.6.1.6, "Application and effect of offset parameters", on page 202.

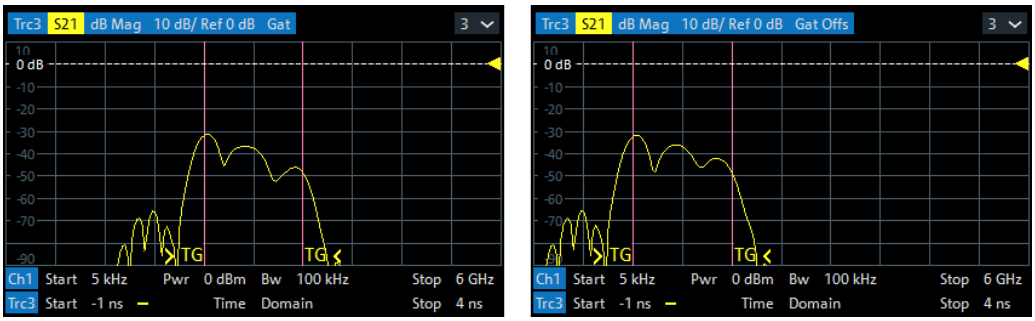
Remote command:
[SENSe<Ch>:]CORRection:EDELay<PhyPt>[:TIME]
[SENSe<Ch>:]CORRection:EDELay<PhyPt>:ELENgth
[SENSe<Ch>:]CORRection:EDELay<PhyPt>:DISTance

Permittivity / Velocity Factor
Defines the permittivity (ε_r) and velocity factor of the dielectric in the transmission line between the reference plane and the DUT. The velocity factor is 1/sqrt(ε_r) and is a measure for the velocity of light in a dielectric with permittivity ε_r relative to the velocity of light in the vacuum (velocity factor < 1). Permittivity and velocity factor are coupled parameters.

See also Section 4.6.1.1, "Definition of offset parameters", on page 198.

Remote command:
[SENSe<Ch>:]CORRection:EDELay<PhyPt>:DIELectric
[SENSe<Ch>:]CORRection:EDELay<PhyPt>:VELocity

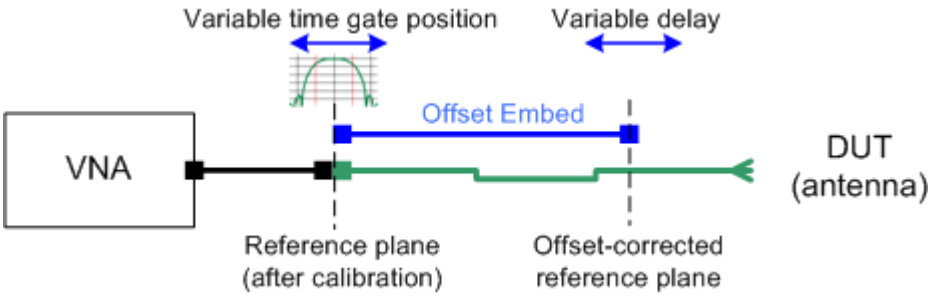
Adjust Time Gate
Activates the operating mode where the time gate is moved in the opposite direction when the "Delay" setting (or any other length offset parameter) is changed. The button is available if a time gate is active (see "Time Gate" on page 364). In time domain, a positive delay shifts the time gate to the left, a negative delay shifts it to the right.



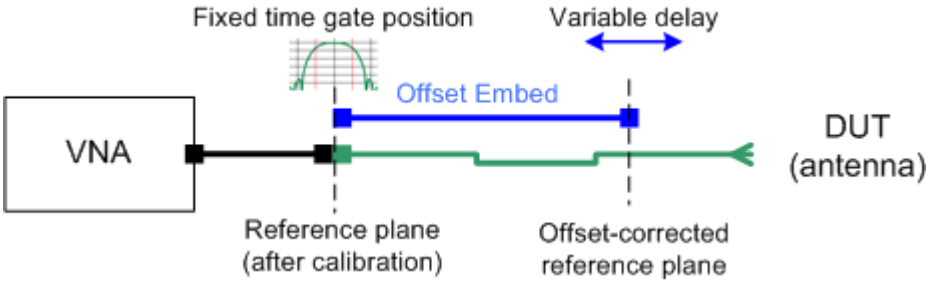
left = no delay
right = delay 1 ns

The position of the time gate is always relative to the end of the offset transmission line. As a consequence, "Adjust Time Gate" allows measurements at variable offset but fixed time gate position.

Example: The impedance of an antenna with possible faults is measured using a time gate and a variable length offset. If "Adjust Time Gate" is off, the time gate is at a constant distance from the offset-corrected reference plane (end of the offset transmission line). Its absolute position is varied along with the length offset.



If "Adjust Time Gate" is on, the time gate is moved to left (right) when the offset-corrected reference plane is moved to the right (left). Its absolute position remains fixed. With this setting, it is possible, e.g., to keep the time gate at the position of the antenna connector while the antenna is measured at different length offsets.



Remote command:
`CALCulate:FILTer[:GATE]:TIME:AOFFset`

Auto Length
Calculates the electrical length offset for the receive port of the active trace. See Section 4.6.1.3, "Auto Length", on page 199.

If "Delay" is the selected trace format, the entire trace is shifted in vertical direction and centered on zero. In phase format, the "Auto Length" corrected trace shows the deviation from linear phase.

Use Evaluation Range... to limit the frequency range for the calculation.

Note that in order to calculate the electrical length offset of port *i*, you have to select a trace that uses port *i* as one of its receive ports. It is **not** relevant for the "Auto Length" function which Port is selected.

If the measured quantity is a ratio, or if it is derived from a ratio, its receiving port is given as the index of the wave quantity in the numerator. If the active trace shows an S-parameter S_{ij}, then "Auto Length" adds a length offset at port i. If the receive port is balanced, then the same offset is applied to both its physical ports.

Remote command:
`[SENSe<Ch>:]CORRection:EDELay:AUTO`

Fixture Compensation
The "Fixture Compensation" button opens the Fixture Compensation dialog.

Evaluation Range...
The "Evaluation Range..." button opens the Evaluation Range dialog that allows you to restrict the frequency range of auto length (and loss) calculations.

5.13.2.2 Fixture Compensation dialog

This dialog allows you to correct the measurement result for the effects of a test fixture.

Access:

- Channel – [Offset Embed] > "Offset" > "Fixture Compensation"
- Channel – [Offset Embed] > "One Way Loss" > "Fixture Compensation"

Background Information
Refer to Section 4.6.1.5, "Fixture Compensation", on page 201.

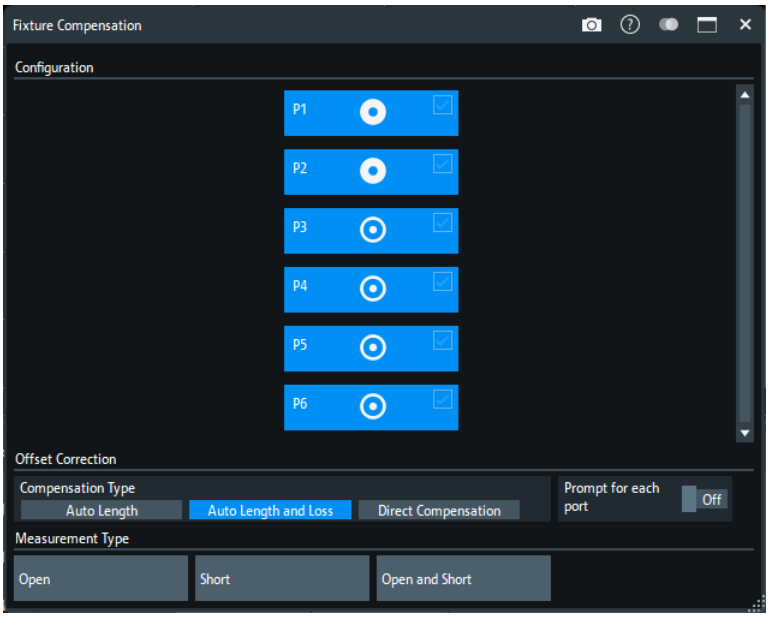


Figure 5-83: Fixture Compensation dialog (for >4 ports)

Ports
Selects the ports for whom fixture compensation data shall be acquired.

Auto Length / Auto Length and Loss
"Auto Length" or "Auto Length and Loss" implies that a global electrical length offset and loss is determined in analogy to the general offset compensation (see Section 4.6.1.3, "Auto Length", on page 199 and Section 4.6.1.4, "Auto Length and Loss", on page 200).

Remote command:
`[SENSe:]CORRection:COLLect:FIXTure:LMParameter:LOSS[:STATe]`

Direct Compensation
With "Direct Compensation", a frequency-dependent transmission factor is calculated; see "Auto Length and Loss vs. Direct Compensation" on page 202.

Remote command:
`[SENSe:]CORRection:COLLect:FIXTure:LMParameter[:STATe]`
`[SENSe<Ch>:]CORRection:OFFSet<PhyPt>:DFComp[:STATe]?`

Prompt for Each Port
Determines how the R&S ZNB performs the sweeps for a given termination type (Open and/or Short; see "Measurement Type" on page 742).

- If disabled, it performs the sweeps for Open/Short without interruption, implicitly assuming that **all** ports are terminated accordingly
- If enabled, it interrupts the data acquisition process after each port, which allows you to modify the test setup (e.g. terminate the next measured port).

Measurement Type
The "Open" , "Short", and "Open and Short" buttons bring up the "Measure Fixture wizard" dialog that guides you through the actual fixture measurement. See "Open/Short vs. Open and Short compensation" on page 202.

Measure Fixture wizard
The "Measure Fixture" dialog guides you through the previously configured fixture compensation measurements.

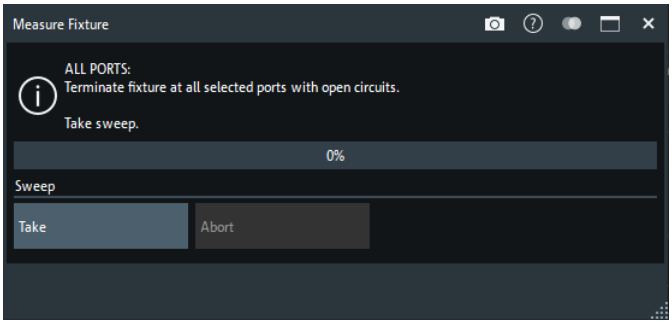
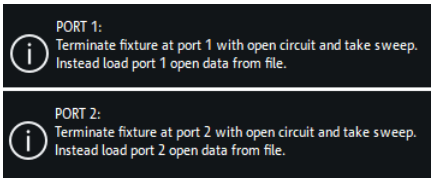


Figure 5-84: Measure Fixture dialog: Auto Length (and Loss)

To acquire the necessary data, proceed as indicated in the information area.

With Prompt for Each Port disabled, "Take" acquires data for all selected ports in one go. Otherwise sweeps are taken port by port.



For "Direct Compensation", it is also possible to save the acquired data to file. In future measurements, you can load these files instead of repeating the data acquisition.

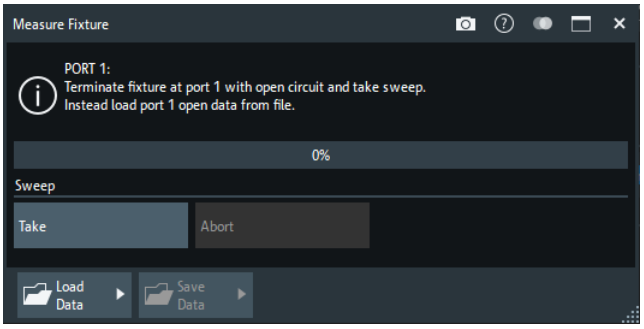


Figure 5-85: Measure Fixture dialog: Direct Compensation (and Prompt for Each Port)

"Direct Compensation" data files are standard trace files, containing reflection parameter traces for the related port and standard:

Table 5-9: Direct Compensation data

Prompt for Each Port	File Type	Description
disabled	csv	One csv trace file per standard, containing reflection traces for all selected ports; see Section 4.4.2.2, "ASCII (*.csv) files", on page 151
enabled	s1p	One 1-port Touchstone file per standard and port (see Section 4.4.2.1, "Touchstone files", on page 144)

Tip: Remote control provides additional flexibility. You can:

- Measure the same ports repeatedly without changing the standards and attribute the results to different channels.
- Calculate the compensation data for different ports, using mixed Open and Short standards.

Refer to the program example for `[SENSe<Ch>:]CORRection:COLLect:FIXTure[:ACQuire]`.

Remote command:

```
[SENSe<Ch>:]CORRection:COLLect:FIXTure:STARt
[SENSe<Ch>:]CORRection:COLLect:FIXTure[:ACQuire]
[SENSe<Ch>:]CORRection:COLLect:FIXTure:SAVE
[SENSe<Ch>:]CORRection:COLLect:FIXTure:EXPort
[SENSe<Ch>:]CORRection:COLLect:FIXTure:IMPort
```

5.13.3 One Way Loss tab

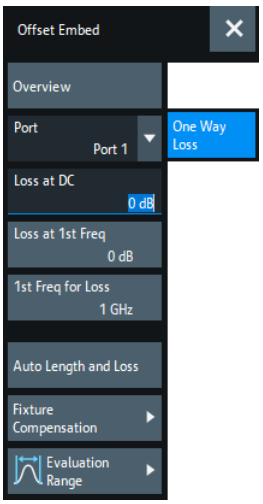
Defines loss parameters for each physical port.

Use the complementary dock widget to activate or deactivate length/loss compensation for selected ports (see Section 5.13.1.3, "Offset panel", on page 705).

Background information

Refer to the following sections.

- Section 4.6, "Offset parameters and de-/embedding", on page 197
- Section 4.6.1.2, "Definition of loss parameters", on page 198
- Section 4.6.1.4, "Auto Length and Loss", on page 200
- Section 4.6.1.5, "Fixture Compensation", on page 201



Overview

This button is available on all de-/embedding-related tabs of the "Offset Embed" softtool. It opens the Overview panel in the Offset Embed dock widget.

Port

Physical test port of the analyzer. You can define independent loss parameters for all ports.

Remote command:
The `<PhyPt>` numeric suffix in the `[SENSe<Ch>:]CORRection:...` commands identifies the physical port.

Loss at DC / Loss at 1st Freq / 1st Freq for Loss

See "Loss at DC / Loss at 1st Freq / 1st Freq for Loss / Use 2nd Freq / Loss at 2nd Freq / 2nd Freq for Loss" on page 706.

Auto Length and Loss

Determines the length offset and one-way loss parameters for the receive port of the active trace. See Section 4.6.1.4, "Auto Length and Loss", on page 200.

Use Evaluation Range... to limit the frequency range for the calculation.

Note:

- If "Auto Length and Loss" is used with a line connected to a test port, the end of the line must be left open.

- To calculate the electrical length offset and loss of port *i*, you have to select a trace that uses port *i* as one of its receive ports. It is **not** relevant for the "Auto Length and Loss" function which Port is selected.

If the measured quantity is a ratio, or if it is derived from a ratio, its receiving port is given as the index of the wave quantity in the numerator. If the active trace shows an S-parameter S_{ij}, then "Auto Length" adds a length offset at port i. If the receive port is balanced, then the same offset and length is applied to both its physical ports.

Remote command:
`[SENSe<Ch>:]CORRection:LOSS:AUTO`

Fixture Compensation
The "Fixture Compensation" button opens the Fixture Compensation dialog.

Evaluation Range...
The "Evaluation Range..." button opens the Evaluation Range dialog that allows you to restrict the frequency range of auto length (and loss) calculations.

5.13.4 Single Ended tab

Allows you to specify 2-port deembedding/embedding networks for each physical port.

Such a network is either defined:

- Via its S-parameters stored in a two-port Touchstone file (`*.s2p`) or
- By selecting a predefined lumped element model and specifying the available parameters (resistances/conductances, capacitances, inductances)

See Section 4.6.2.3, "Circuit models for 2-port networks", on page 205.

Use the complementary dock widget to activate or deactivate dembedding/embedding for selected ports (see Section 5.13.1.4, "Single Ended panel", on page 707).

Background information
Refer to the section Section 4.6.2.10, "Combining several de-/embedding networks", on page 214.

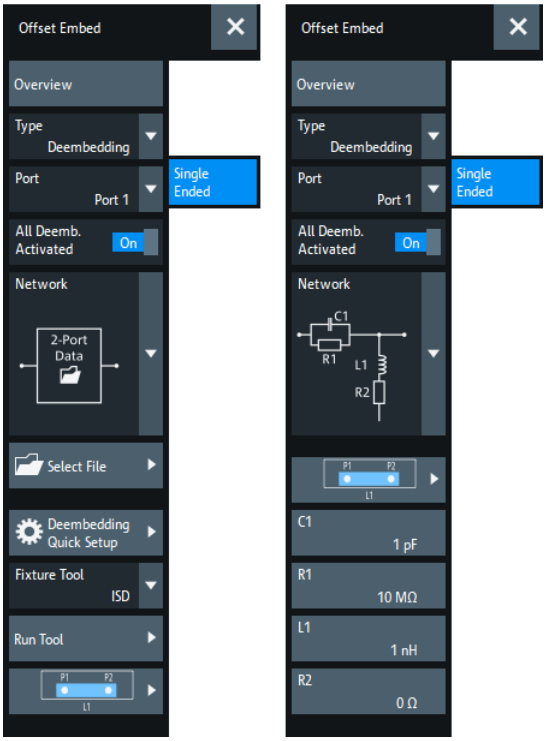


Figure 5-86: Offset Embed > Single Ended softtool tab

left = 2-port data file selected for deembedding
right = lumped element model selected for embedding

If the "Fixture Simulator" is disabled for the related channel (see "Fixture Simulator" on page 672), this tab is inactive, i.e. all controls except the "Overview" button are grayed out.

Overview
This button is available on all de-/embedding-related tabs of the "Offset Embed" softtool. It opens the Overview panel in the Offset Embed dock widget.

Type
Switches between "Deembedding" and "Embedding" network definition.

Port
Physical port. The transformation networks are defined such that the analyzer is connected to the left of the circuit while the DUT is connected to the right side. You can define independent transformation networks for all ports.

Remote command:
The `<PhyPt>` numeric suffix in the embedding/deembedding commands identifies the physical port; see e.g. `CALCulate<Ch>:TRANsform:VNETworks:SENDed:DEEMbedding<PhyPt>[:STATe]` or `CALCulate<Ch>:TRANsform:VNETworks:SENDed:EMBedding<PhyPt>[:STATe]`.

All Deembedding Activated
Deactivates or activates all deembeddings (single-ended, balanced, port set, ground loop) configured for the active channel.

This control is only visible if Type = "Deembedding".

Remote command:
```
CALCulate<Ch>:TRANsform:VNETworks:ACTivateall[:STATe]
```

Network
The graphical list contains all available 2-port networks:
- The symbol selects "no network" and disables single-ended de-/embedding.
- The "2-Port Data" network is defined by an s2p file (see Select File). For deembedding, the s2p file can also be generated by a third-party fixture modeling tool or by the delta cal method (see "Fixture Tool/Run Tool" on page 748).
- The remaining networks are defined by lumped elements. Their parameters are displayed below the graphical list.

Tip: Drag and drop the network symbols in horizontal or vertical direction to switch to the next symbol. The lumped elements are numbered from top to bottom.

Remote command:
```
CALCulate<Ch>:TRANsform:VNETworks:SENDed:DEEMbedding<PhyPt>[:STATe]
CALCulate<Ch>:TRANsform:VNETworks:SENDed:DEEMbedding<PhyPt>:TNDefinition
CALCulate<Ch>:TRANsform:VNETworks:SENDed:DEEMbedding<PhyPt>:PARameters:C<Cmp>
CALCulate<Ch>:TRANsform:VNETworks:SENDed:DEEMbedding<PhyPt>:PARameters:G<Cmp>
CALCulate<Ch>:TRANsform:VNETworks:SENDed:DEEMbedding<PhyPt>:PARameters:L<Cmp>
CALCulate<Ch>:TRANsform:VNETworks:SENDed:DEEMbedding<PhyPt>:PARameters:R<Cmp>
CALCulate<Ch>:TRANsform:VNETworks:SENDed:EMBedding<PhyPt>[:STATe]
CALCulate<Ch>:TRANsform:VNETworks:SENDed:EMBedding<PhyPt>:TNDefinition
CALCulate<Ch>:TRANsform:VNETworks:SENDed:EMBedding<PhyPt>:PARameters:C<Cmp>
CALCulate<Ch>:TRANsform:VNETworks:SENDed:EMBedding<PhyPt>:PARameters:G<Cmp>
CALCulate<Ch>:TRANsform:VNETworks:SENDed:EMBedding<PhyPt>:PARameters:L<Cmp>
CALCulate<Ch>:TRANsform:VNETworks:SENDed:EMBedding<PhyPt>:PARameters:R<Cmp>
```

Select File
"Select File" is enabled as long as the "2-Port Data" network is selected. This network is defined by its S-parameters stored in a two-port Touchstone file (`*.s2p`). No additional parameters are required.

A click or tap opens the Open de-/embedding file dialog.

Note: The loaded file is stored in the active recall set. Recall sets contain the full de-/embedding data so that they can be transferred to other instruments.

Remote command:
`MMEMory:LOAD:VNETworks<Ch>:SENDed:DEEMbedding<PhyPt>`
`MMEMory:LOAD:VNETworks<Ch>:SENDed:EMBedding<PhyPt>`

Deembedding Quick Setup...
Opens the Deembedding Quick Setup dialog.

Fixture Tool/Run Tool
Only available for deembedding 2-port data networks.

Allows you to select and run a third-party fixture modeling tool (see Section 4.6.2.9, "Fixture modeling and deembedding", on page 212) and to use its results for single-ended deembedding.

If the selected fixture tool is not installed on the instrument, the "Run Tool" button is disabled/grayed out. Otherwise it opens the Fixture modeling tool panel that guides you through the fixture modeling.

Note: Delta Cal only works for channels without balanced ports. If the active channel contains a balanced port, "Run Tool" is disabled.

Remote command:
`CALCulate:FMODel:DEASsistant:TOOL`

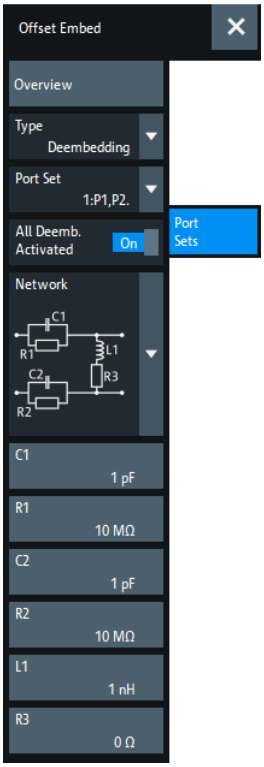
5.13.5 Port Sets tab

Selects transformation networks for deembedding/embedding arbitrary port sets, defines their parameters, assigns them to a port set and enables embedding.

Use the complementary dock widget to create the required port sets and to activate or deactivate dembedding/embedding for selected port sets (see Section 5.13.1.5, "Port Sets panel", on page 708).

Background information

Refer to Section 4.6.2.5, "Port pair de-/embedding", on page 208 and Section 4.6.2.6, "Port set de-/embedding", on page 209.



If the "Fixture Simulator" is disabled for the related channel (see "Fixture Simulator" on page 672), this tab is inactive, i.e. all controls except the "Overview" button are grayed out.

Overview
See "Overview" on page 738.

Type
Switches between "Deembedding" and "Embedding" network definition.

Port Set
Port sets, defined in the complementary Port Sets panel dock widget panel. The transformation networks are defined such that the physical analyzer test ports are connected to the left of the circuit; the DUT ports are on the right side. You can define independent transformation networks for all port sets.

The port set number, i.e. the number at the beginning of each "Port Set" item, corresponds to the position of the port set in the Port Sets panel.

Remote command:
The port set number corresponds to the `<ListId>` numeric suffix in the port set de-/embedding commands; see e.g. `CALCulate<Ch>:TRANsform:VNETworks:PPAir:DEEMbedding<ListId>[:STATe]`.

All Deembedding Activated
Deactivates or activates all deembeddings (single-ended, balanced, port set, ground loop) configured for the active channel.

This control is only visible if Type = "Deembedding".

Remote command:
CALCulate<Ch>:TRANsform:VNETworks:ACTivateall[:STATe]

Network
Depending on the size *m* of the selected port set, the graphical list contains all available 2*m*-port deembedding/embedding networks.

Note: For port *pairs* (m=2) the deembedding/embedding network can be defined either via lumped element models (in combination with `s2p` Touchstone files) or via a `s4p` Touchstone file (see Section 4.6.2.5, "Port pair de-/embedding", on page 208). For m≥3, there are no predefined lumped element models available; the deembedding/embedding network has to be defined via an s<2m>p Touchstone file.

The  symbol selects "no network" and disables de-/embedding for the selected port set. The "D1" and "D2" networks are defined by imported S-parameter data; see D1, D2.

For 4-port networks that are (partly) defined by lumped elements, the lumped element parameters are displayed below the graphical network list. See Section 4.6.2.4, "Circuit models for 4-port networks", on page 207.

Tip:
- Drag and drop the network symbols in horizontal or vertical direction to switch to the next symbol. The lumped elements and S-parameter networks ("D1", "D2") are numbered from top to bottom.
- Use the Conductance in Embedding Networks switch to change from resistances in "Capacitor in parallel with resistor" circuit blocks to conductances and vice versa.

Remote command:
CALCulate<Ch>:TRANsform:VNETworks:PPAir:DEEMbedding<ListId>:TNDefinition
CALCulate<Ch>:TRANsform:VNETworks:PPAir:DEEMbedding<ListId>:PARameters:C<1|2|3>
CALCulate<Ch>:TRANsform:VNETworks:PPAir:DEEMbedding<ListId>:PARameters:L<1|2|3>
CALCulate<Ch>:TRANsform:VNETworks:PPAir:DEEMbedding<ListId>:PARameters:R<1|2|3>
CALCulate<Ch>:TRANsform:VNETworks:PPAir:DEEMbedding<ListId>:PARameters:G<1|2|3>
CALCulate<Ch>:TRANsform:VNETworks:PPAir:EMBedding<ListId>:TNDefinition
CALCulate<Ch>:TRANsform:VNETworks:PPAir:EMBedding<ListId>:PARameters:C<1|2|3>
CALCulate<Ch>:TRANsform:VNETworks:PPAir:EMBedding<ListId>:PARameters:L<1|2|3>
CALCulate<Ch>:TRANsform:VNETworks:PPAir:EMBedding<ListId>:PARameters:R<1|2|3>
CALCulate<Ch>:TRANsform:VNETworks:PPAir:EMBedding<ListId>:PARameters:G<1|2|3>

D1, D2
The "D1" (and "D2") buttons are enabled as long as the selected deembedding/embedding network is defined using Touchstone files.

A click or tap opens the Open de-/embedding file dialog.

Note: The loaded file is stored in the active recall set. Recall sets contain the full de-/embedding data so that they can be transferred to other instruments.

Remote command:
`MMEMory:LOAD:VNETworks<Ch>:PPAir:DEEMbedding<ListId>`
`MMEMory:LOAD:VNETworks<Ch>:PPAir:EMBedding<ListId>`

5.13.6 Balanced tab

Selects 4-port transformation networks for balanced port deembedding/embedding, defines their parameters, assigns them to a balanced port and enables embedding.

Use the complementary dock widget to create Balanced panel and to activate or deactivate dembedding/embedding for selected balanced ports (see Section 5.13.1.6, "Balanced panel", on page 710).



Background information
Refer to the section Section 4.6.2.10, "Combining several de-/embedding networks", on page 214.

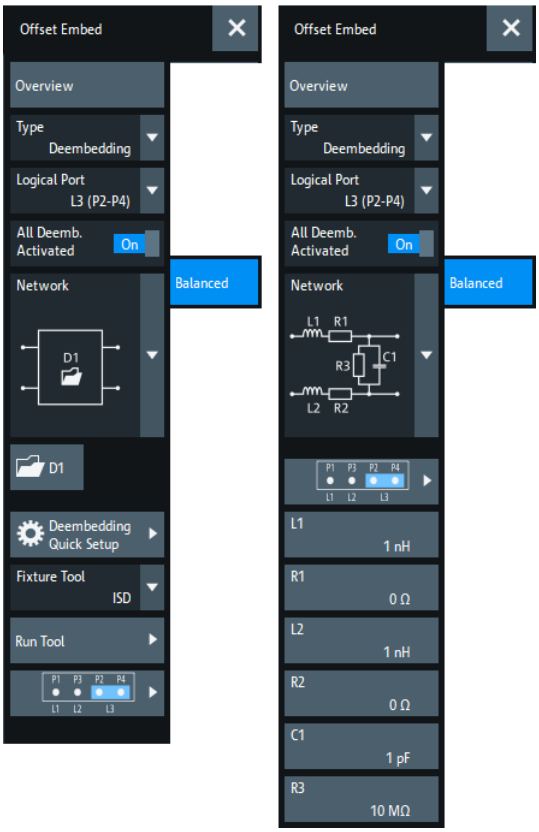


Figure 5-87: Offset Embed > Balanced softtool tab

left = 4-port data file selected for deembedding
right = other network selected for embedding

If the "Fixture Simulator" is disabled for the related channel (see "Fixture Simulator" on page 672), this tab is inactive, i.e. all controls except the "Overview" button are grayed out.

Overview
This button is available on all de-/embedding-related tabs of the "Offset Embed" softtool. It opens the Overview panel in the Offset Embed dock widget.

Type
Switches between "Deembedding" and "Embedding" network definition.

Logical Port
Logical analyzer port, as defined in the "Balanced Ports" configuration. The transformation networks are defined such that the physical analyzer test ports are connected to the left of the circuit; the DUT ports are on the right side.

You can define independent transformation networks for all balanced ports.

Remote command:
The <LogPt> numeric suffix in the embedding/deembedding commands identifies the logical port; see e.g. CALCulate<Ch>:TRANsform:VNETworks:BALanced:EMBedding<LogPt>[:STATe].

All Deembedding Activated
Deactivates or activates all deembeddings (single-ended, balanced, port set, ground loop) configured for the active channel.

This control is only visible if Type = "Deembedding".

Remote command:
CALCulate<Ch>:TRANsform:VNETworks:ACTivateall[:STATe]

Network
The graphical list contains all available 4-port networks (see Section 4.6.2.4, "Circuit models for 4-port networks", on page 207).

The symbol selects "no network" and disables deembedding/embedding for the selected balanced port.

The 4-port data network (symbol "D1" only) is defined by an s4p file (see D1, D2). For deembedding, the s4p file can also be generated by a third-party fixture modeling tool (see "Fixture Tool/Run Tool" on page 748).

Other 2-port data-subnetworks (symbols "D1" and "D2") are defined by s2p files (see Network).

The parameters of lumped elements are displayed below the graphical list.

Tip:
- Drag and drop the network symbols in horizontal or vertical direction to switch to the next symbol. The lumped elements and S-parameter networks ("D1", "D2") are numbered from top to bottom.
- Use the Conductance in Embedding Networks switch to change from resistances in "Capacitor in parallel with resistor" circuit blocks to conductances and vice versa.

Remote command:
CALCulate<Ch>:TRANsform:VNETworks:BALanced:DEEMbedding<LogPt>:TNDefinition
CALCulate<Ch>:TRANsform:VNETworks:BALanced:DEEMbedding<LogPt>:PARameters:C<Cmp>
CALCulate<Ch>:TRANsform:VNETworks:BALanced:DEEMbedding<LogPt>:PARameters:L<Cmp>
CALCulate<Ch>:TRANsform:VNETworks:BALanced:DEEMbedding<LogPt>:PARameters:R<Cmp>
CALCulate<Ch>:TRANsform:VNETworks:BALanced:DEEMbedding<LogPt>:PARameters:G<Cmp>
CALCulate<Ch>:TRANsform:VNETworks:BALanced:EMBedding<LogPt>:TNDefinition
CALCulate<Ch>:TRANsform:VNETworks:BALanced:EMBedding<LogPt>:PARameters:C<Cmp>
CALCulate<Ch>:TRANsform:VNETworks:BALanced:EMBedding<LogPt>:PARameters:L<Cmp>

CALCulate<Ch>:TRANsform:VNETworks:BALanced:EMBedding<LogPt>:PARameters:R<Cmp>
CALCulate<Ch>:TRANsform:VNETworks:BALanced:EMBedding<LogPt>:PARameters:G<Cmp>

D1, D2
The "D1" and "D2" buttons are enabled as long as the selected Network comprises subnetworks that are defined via two-port or four-port Touchstone files (*.s2p, *.s4p).

A click or tap opens the Open de-/embedding file dialog.

Note: The loaded file is stored in the active recall set. Recall sets contain the full de-/embedding data so that they can be transferred to other instruments.

Remote command:
MMEMory:LOAD:VNETworks<Ch>:BALanced:DEEMbedding<LogPt>
MMEMory:LOAD:VNETworks<Ch>:BALanced:EMBedding<LogPt>

Deembedding Quick Setup...
Opens the Deembedding Quick Setup dialog.

Fixture Tool/Run Tool
Only available for deembedding 4-port data networks.

Allows you to select and run a third-party fixture modeling tool (see Section 4.6.2.9, "Fixture modeling and deembedding", on page 212) and to use its results for single-ended deembedding.

If the selected fixture modeling tool is not installed on the instrument, the "Run Tool" button is disabled/grayed out. Otherwise it opens the Fixture modeling tool panel that guides you through the fixture modeling.

Note:
- Define at least two Balanced panel before running the modeling tool.
- Delta Cal does not work for balanced ports and hence it is not offered as "Fixture Tool" here.

5.13.7 Ground Loop tab

Allows you to specify a 1-port ground loop de-/embedding network.

Such a network is either defined:

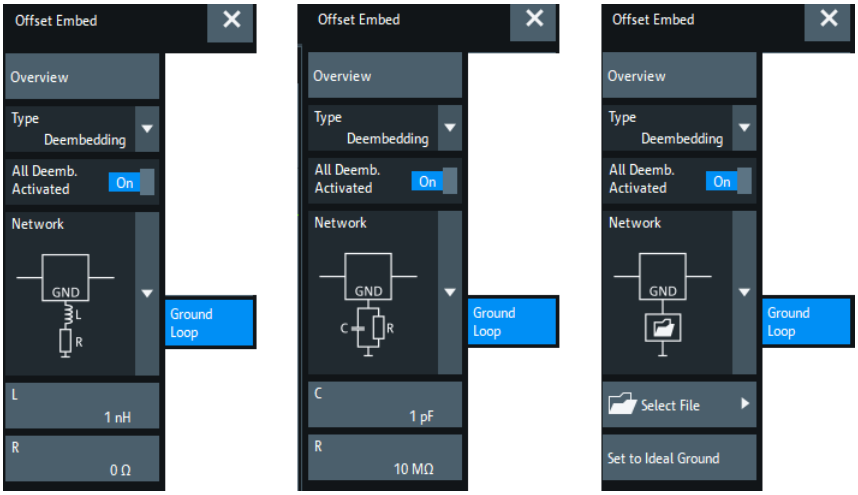
- Via its S-parameter stored in a one-port Touchstone file (*.s1p) or
- By selecting a predefined lumped element model (Shunt L or Shunt C) and specifying the available parameters (resistance/inductance or resistance/capacitance)

Use the complementary dock widget to activate or deactivate ground loop deemmbedding and embedding (see Section 5.13.1.7, "Ground Loop panel", on page 711).

If multiple port groups (DUTs) are configured and Ground Loop per Port Group is enabled, use the dock widget to select the port group whose ground loop network you want to configure.

Background information

Refer to Section 4.6.2.7, "Ground loop de-/embedding", on page 211.



If the "Fixture Simulator" is disabled for the related channel (see "Fixture Simulator" on page 672), this tab is inactive, i.e. all controls except the "Overview" button are grayed out.

Overview

This button is available on all de-/embedding-related tabs of the "Offset Embed" softtool. It opens the Overview panel in the Offset Embed dock widget.

Type

Switches between "Deembedding" and "Embedding" network definition.

All Deembedding Activated

Deactivates or activates all deembeddings (single-ended, balanced, port set, ground loop) configured for the active channel.

This control is only visible if Type = "Deembedding".

Remote command:

`CALCulate<Ch>:TRANsform:VNETworks:ACTivateall[:STATe]`

Network

The graphical list contains all available 1-port networks:

The symbol selects "no network" and disables ground loop de-/embedding.

The "1-Port Data" network is defined by imported S-parameter data; see Select File.

The remaining networks (Shunt L and Shunt C) are defined by lumped elements whose parameters are displayed below the graphical list.

Tip:

- Drag and drop the network symbols in horizontal or vertical direction to switch to the next symbol.

- Use the Conductance in Embedding Networks switch to change from resistances in "Capacitor in parallel with resistor" model to conductances and vice versa.

Remote command:

```
CALCulate<Ch>:TRANsform:VNETworks:GLOop:DEEMbedding<group>:
TNDefinition
CALCulate<Ch>:TRANsform:VNETworks:GLOop:DEEMbedding<group>:
PARameters:C
CALCulate<Ch>:TRANsform:VNETworks:GLOop:DEEMbedding<group>:
PARameters:L
CALCulate<Ch>:TRANsform:VNETworks:GLOop:DEEMbedding<group>:
PARameters:R
CALCulate<Ch>:TRANsform:VNETworks:GLOop:DEEMbedding<group>:
PARameters:G
CALCulate<Ch>:TRANsform:VNETworks:GLOop:EMBedding<group>:
TNDefinition
CALCulate<Ch>:TRANsform:VNETworks:GLOop:EMBedding<group>:
PARameters:C
CALCulate<Ch>:TRANsform:VNETworks:GLOop:EMBedding<group>:
PARameters:L
CALCulate<Ch>:TRANsform:VNETworks:GLOop:EMBedding<group>:
PARameters:R
CALCulate<Ch>:TRANsform:VNETworks:GLOop:EMBedding<group>:
PARameters:G
```

Select File

"Select File" is enabled as long as the "1-Port Data" network is selected. This network is defined by its S-parameters stored in a one-port Touchstone file (`*.s1p`). No additional parameters are required.

A click or tap opens the Open de-/embedding file dialog.

Note: The loaded file is stored in the active recall set. Recall sets contain the full de-/embedding data so that they can be transferred to other instruments.

Remote command:

```
MMEMory:LOAD:VNETworks<Ch>:GLOop:DEEMbedding<group>
MMEMory:LOAD:VNETworks<Ch>:GLOop:EMBedding<group>
```

Set to Ideal Ground

This function is enabled as long as the 1-Port data network is active. An imported S-parameter set is replaced by the S-parameters of an ideal through connection, which eliminates the transformation network.

Ground Loop per Port Group

If selected, each port group (= DUT) can have its own ground loop deembedding and embedding networks. Otherwise the same de-/embedding network is used for all active ports.

The control is only visible if the instrument supports multiple DUTs, which is the case for R&S ZNBT and 4-port R&S ZNB with 2 internal sources.

Remote command:
`CALCulate<Ch>:TRANsform:VNETworks:GLOop:GROup`

5.13.8 Differential Match tab

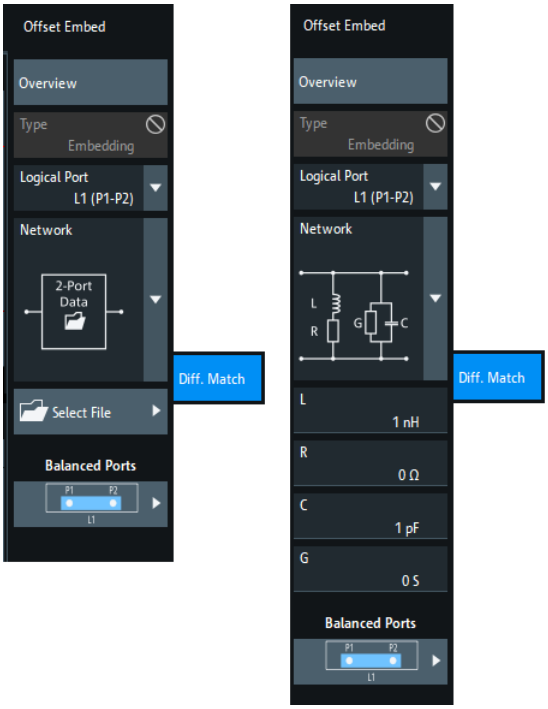
Allows you to specify a 2-port embedding network for the differential mode of a balanced port.

Such a network is either defined:

- Via its S-parameter stored in a two-port Touchstone file (`*.s2p`)
- By specifying the parameters of a "Shunt L, Shunt C" lumped element model

Use the complementary dock widget to access the balanced port configuration and to activate or deactivate "Differential Match" embedding for selected balanced ports (see Section 5.13.1.9, "Differential Match panel", on page 713).

Background information
Refer to Section 4.6.2.8, "Differential match embedding", on page 212.



If the "Fixture Simulator" is disabled for the related channel (see "Fixture Simulator" on page 672), this tab is inactive, i.e. all controls except the "Overview" button are grayed out.

Overview
This button is available on all de-/embedding-related tabs of the "Offset Embed" softtool. It opens the Overview panel in the Offset Embed dock widget.

Type
Currently only Differential Match"Embedding" is supported.

Logical Port
Logical analyzer port, as defined in the "Balanced Ports" configuration. The transformation networks are defined such that the physical analyzer test ports are connected to the left of the circuit; the DUT ports are on the right side. You can define independent embedding networks for all balanced ports.

Remote command:
The `<LogPt>` numeric suffix in the embedding/deembedding commands identifies the logical port; see e.g. `CALCulate<Ch>:TRANsform:VNETworks:DIFFerential:EMBedding<LogPt>[:STATe]`.

Network
The graphical list contains the available 2-port networks for Differential Match embedding:

- The symbol selects "no network" and disables differential match embedding for the selected balanced port.
- The "2-Port Data" network is defined by imported S-parameter data; see Select File
- The "Shunt L, Shunt C" network is defined by lumped elements whose parameters are displayed below the graphical list.

Tip: Drag and drop the network symbols in horizontal or vertical direction to switch to the next symbol.

Remote command:
`CALCulate<Ch>:TRANsform:VNETworks:DIFFerential:EMBedding<LogPt>:TNDefinition`
`CALCulate<Ch>:TRANsform:VNETworks:DIFFerential:EMBedding<LogPt>:PARameters:L<Cmp>`
`CALCulate<Ch>:TRANsform:VNETworks:DIFFerential:EMBedding<LogPt>:PARameters:R<Cmp>`
`CALCulate<Ch>:TRANsform:VNETworks:DIFFerential:EMBedding<LogPt>:PARameters:C<Cmp>`
`CALCulate<Ch>:TRANsform:VNETworks:DIFFerential:EMBedding<LogPt>:PARameters:G<Cmp>`

Select File
"Select File" is enabled as long as the "2-Port Data" network is selected. This network is defined by its S-parameters stored in a two-port Touchstone file (`*.s2p`). No additional parameters are required.

A click or tap opens the Open de-/embedding file dialog.

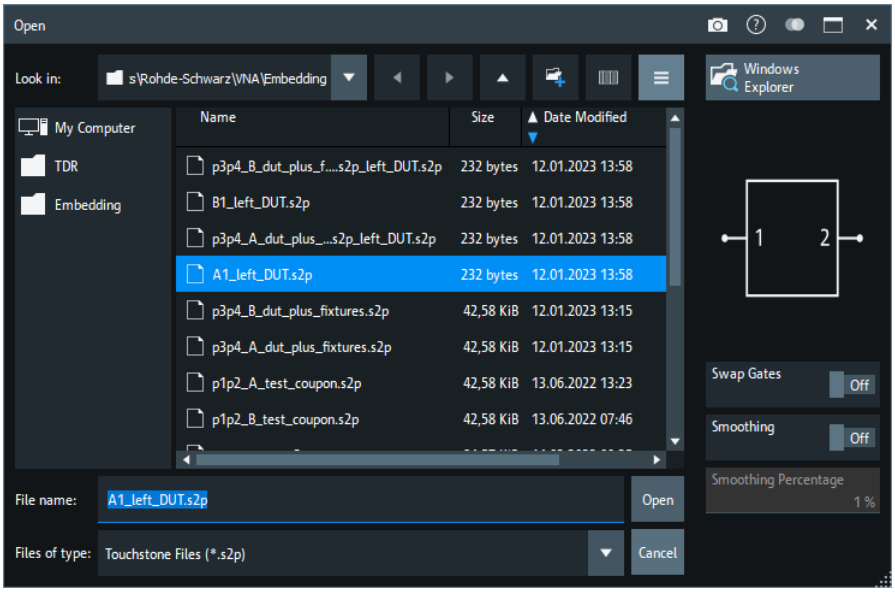
Note: The loaded file is stored in the active recall set. Recall sets contain the full de-/embedding data so that they can be transferred to other instruments.

Remote command:
`MMEMory:LOAD:VNETworks<Ch>:DIFFerential:EMBedding<LogPt>`

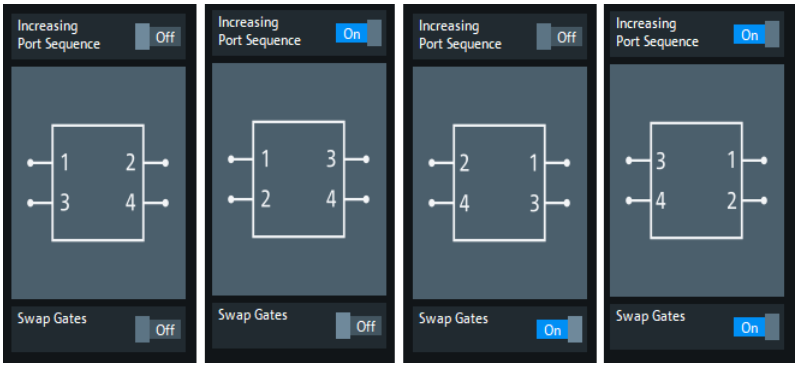
5.13.9 Open de-/embedding file dialog

Allows you to pick a Touchstone file for de-/embedding.

The additional functionality offered on the right-hand side of the dialog depends on whether the file is picked for deembedding or embedding, and on the number of physical DUT ports to be de-/embedded.



Increasing Port Sequence/Swap Gates
When loading a touchstone file, the analyzer by default assumes odd ports left (VNA side), even ports right (DUT side). However, it is also possible to instruct the analyzer to assume an "Increasing Port Sequence", i.e. that the low port numbers are on the VNA side. Furthermore it is possible to "Swap Gates", i.e. to instruct the analyzer to swap the VNA and DUT side.



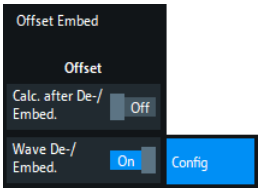
Smoothing/Smoothing Percentage
Only available for deembedding types Single Ended, Port Sets, and Balanced.

For high loss scenarios, smoothing the deembedding data (i.e. the magnitude of the S-parameters) can reduce the appearance of noise in traces.

The "Smoothing Percentage" defines how many trace values are averaged.

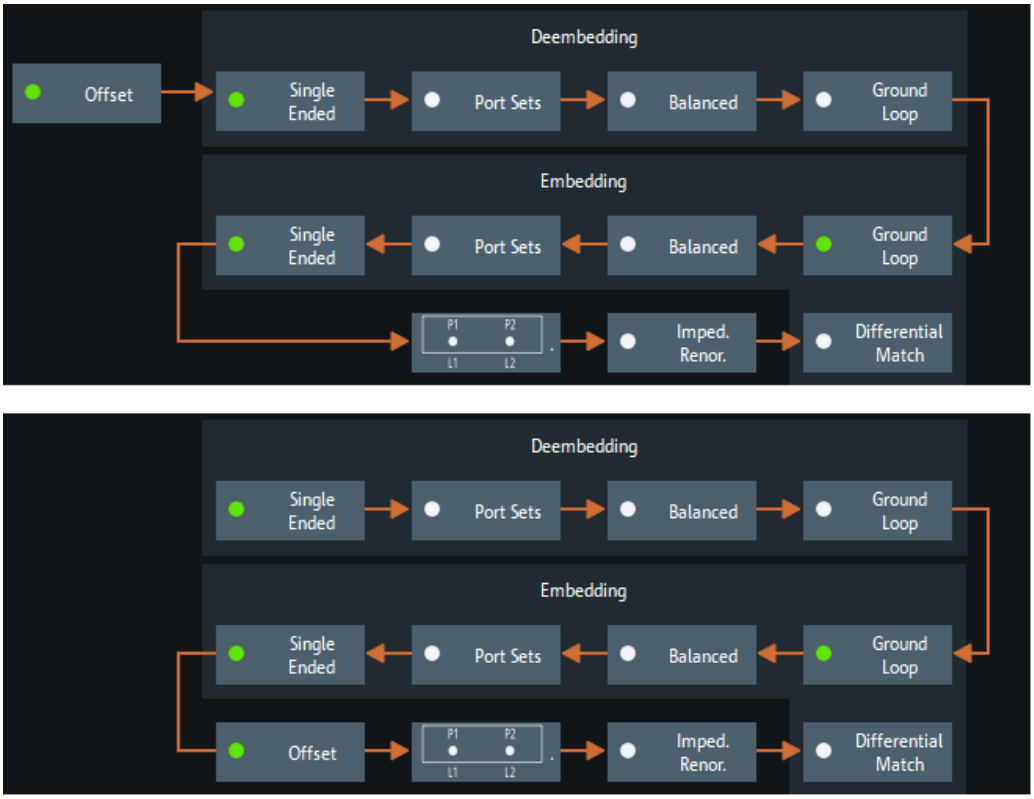
5.13.10 Config tab

The "Config" tab provides means to configure the overall "Offset Embed" calculation.



Offset > Calculate after De-/Embed.
Changes the position of the "Offset" calculation in the "Offset Embed" calculation chain.

If disabled (default), the offset is calculated before de-/embedding. If enabled, it is calculated after de-/embedding.



Top = Default offset calculation
Bottom = Offset calculation after de-/embedding (GUI mockup)

Remote command:
`[SENSe:]CORRection:EDELay:VNETwork`

Offset > Wave De-/Embed.
If enabled, the firmware uses the new, wave-based de-/embedding calculation (default). Otherwise, the legacy S parameter-based calculation is used. See Section 4.1.5, "Data flow", on page 89.

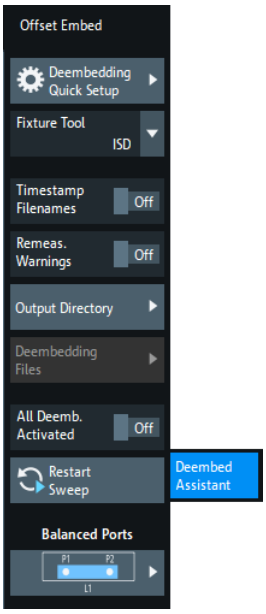
Remote command:
CALCulate<Ch>:TRANsform:VNETworks:WAVes

5.13.11 Deembed Assistant tab

The "Deembed Assistant" softtool tab and the related Deembedding Assistant panel simplify common fixture deembedding scenarios.

Background information
Refer to Section 4.6.2.9, "Fixture modeling and deembedding", on page 212.

5.13.11.1 Controls on the Deembed Assistant tab



Overview
This button is available on all de-/embedding-related tabs of the "Offset Embed" softtool. It opens the Overview panel in the Offset Embed dock widget.

Deembedding Quick Setup...
Opens the Deembedding Quick Setup dialog.

Fixture Tool
The "Fixture Tool" combo-box allows you to select the third-party fixture modeling tool to be used with the deembedding assistant (see Section 4.6.2.9, "Fixture modeling and deembedding", on page 212). After having selected the tool, the Deembedding Assistant panel is shown.

Remote command:
CALCulate:FMODel:DEASsistant:TOOL

Timestamp Filenames
If "Timestamp Filenames" is enabled, the names of subsequently generated "Test Coupon" and "DUT + Test Fixture" files are prefixed with the current date and time.

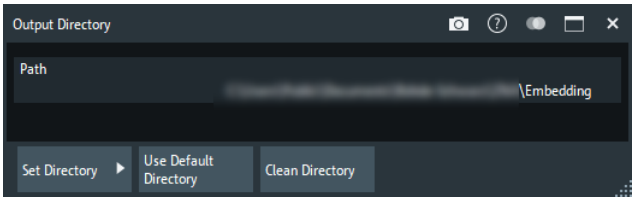
Remote command:
`CALCulate:FMODel:REName`

Show Remeasure Warnings
If enabled, the "Coupon" and "DUT + Test Fixture" measurements raise warnings if the measurement is repeated with different ports.

This setting is global.

Remote command:
n.a.

Output Directory...
Opens a dialog that allows you to specify the directory where the fixture modeling tools store their data.



Default is `C:\Users\Public\Documents\Rohde-Schwarz\Vna\Embedding`.

Remote command:
`CALCulate:FMODel:DIRectory`
`CALCulate:FMODel:DIRectory:DEFault`
`CALCulate:FMODel:DIRectory:DEFault:CLEar`

Deembedding Files...
This button is only enabled if the coupon data provided to the deembedding tool was at least partly loaded from file. See "Page 2: Measure / Load and Apply" on page 734.

A section "Coupon B" is only available if you disable "Use Same Coupon Left and Right" on page 733 and specify a file instead of measuring "Coupon B".

For "DeltaCal" there are no such deembedding files.

Remote command:
`CALCulate:FMODel:DEASsistant:COUPon:LEFT:FILE`
`CALCulate:FMODel:DEASsistant:COUPon:RIGHt:FILE`

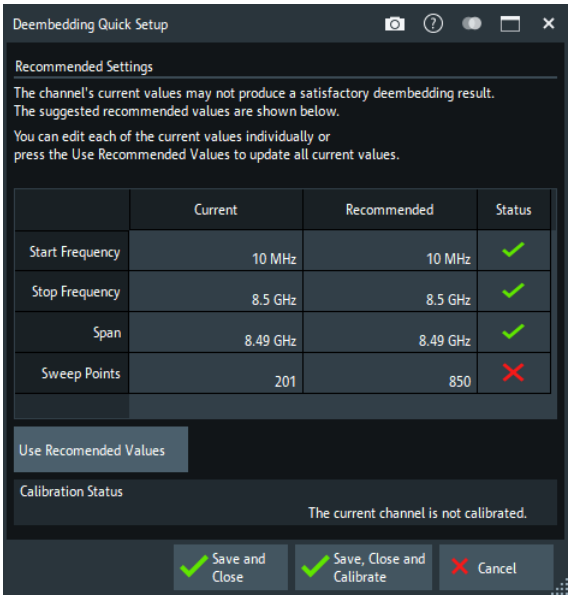
All Deembedding Activated
Deactivates or activates all deembeddings (single-ended, balanced, port set, ground loop) for the active channel, if any.

Remote command:
n.a.

Balanced Ports
Opens the Balanced Ports dialog.

5.13.11.2 Deembedding Quick Setup dialog

Guides you in choosing channel settings that permit accurate modeling with the selected fixture tool (ISD, SFD, or EZD).



The dialog contrasts actual and recommended settings, and highlights the values we recommend adjusting. The values in the "Current" column can be edited; if you modify one of them, the recommended values are updated according to the new value.

The "Current" values are not applied unless you use one of the "Save..." buttons to exit the dialog (and proceed with or without calibration). Select"Cancel" to leave the settings unchanged.

Recommended/Use Recommended Values
Some fixture modeling tools need a wider frequency range, some need more sweep points, some need a harmonic grid to yield better results. The "Recommended" settings represent the experience the Rohde & Schwarz experts have made with the supported fixture modeling tools. However, you are not bound to our recommendations and can use different settings instead.

Select "Use Recommended Values" to adopt the Recommended/Use Recommended Values values with a single tap or click.

Once the deembedding is calculated, you can change the related channel settings according to your measurement task.

Remote command:
`CALCulate:FMODel:QSETup`

5.13.11.3 Deembedding Assistant panel

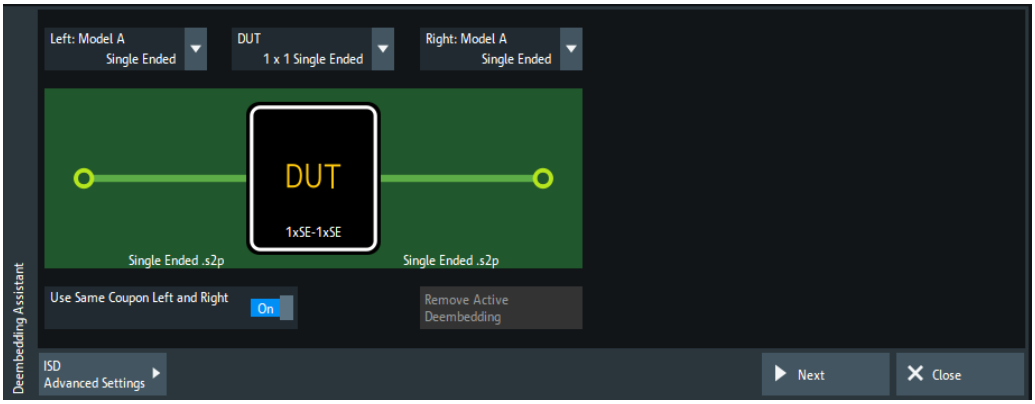
The two pages of the "Offset Embed" dock widget's "Deembedding Assistant" panel allow you to model your DUT and its test fixture, to collect the input data for the selected fixture tool, to run the tool, and to apply the calculated deembedding.

Access: Channel – [Offset Embed] > "Deembed Assistant" tab

(Click the tab to reopen)

Page 1: Topology

The first page of the "Deembedding Assistant" defines the deembedding topology. It allows you to describe your device under test (DUT) in terms of inputs, outputs, and the test coupons required to model the DUT's fixture.



Depending on the selected fixture tool, it narrows down the selectable DUT types and test coupon models.

For DeltaCal deembedding, only single-ended DUTs and fixture models are supported.

DUT

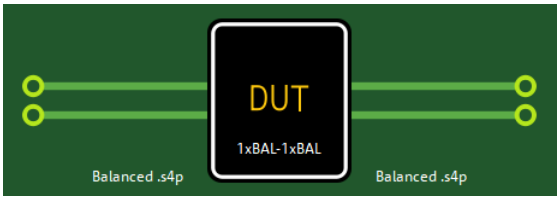
The first step in setting up a new deembedding with the "Deembedding Assistant" is to select the type of DUT. Depending on the number of available test/DUT ports and the selected fixture tool, choose one of the following DUT types:

Table 5-10: DUT types

DUT type	Description	ISD only
"1 x 1 Single Ended"	1 single-ended input, 1 single-ended output	☐
"1 x Single Ended"	1 single-ended input, no output	☐
"1 x 1 Balanced"	1 balanced input, 1 balanced output	☐
"1 x Balanced"	1 balanced input, no output	☐
"1 x Balanced, 1 x Single Ended"	1 balanced input, 1 single-ended output	☑
"2 x 1 Single Ended"	2 single-ended inputs, 1 single-ended output	☑
"2 x 2 Single Ended"	2 single-ended inputs, 2 single-ended outputs	☑
"n x Single Ended"	n single-ended inputs, no outputs	☑
"n x m Single Ended"	n single-ended inputs, m single-ended outputs	☑
"n x Balanced"	n balanced inputs, no outputs	☑

DUT type	Description	ISD only
"n x m Balanced"	n balanced inputs, m balanced outputs	☑
"n x n Multiport"		☑

The dock widget presents a graphical representation of the selected "DUT" and coupon models, e.g. for a "1 x 1 Balanced" DUT witch "Balanced" left and right models:



Remote command:

```
CALCulate:FMODel:DEASsistant:DUT:TYPE
CALCulate:FMODel:DEASsistant:PRESet
```

Left: Model A / Right: Model A|B

Allows you to select a coupon type, i.e. to define how the coupling at the left side (lead-in) and right side (lead-out) of the DUT is modeled.

If you select the same model for the left and right side and enable "Use Same Coupon Left and Right" on page 733, then also the right side is labeled with "Model A" and in the measurement phase only the data of a single coupon ("Coupon A") must be measured or loaded from file.

"Single Ended" This coupon model is appropriate for one or more single-ended inputs (outputs), where:

- All inputs (outputs) use the same coupon model and
- There is no interaction between inputs (outputs) if more than one is present on the DUT

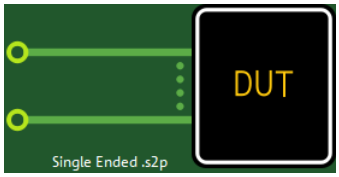


Figure 5-88: n single-ended input ports (uncoupled)

"Balanced" This coupon type is appropriate for one or more balanced inputs (outputs), if:

- All inputs (outputs) use the same coupon model and
- There is no interaction between inputs (outputs) if more than one is present on the DUT

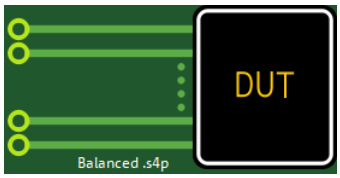


Figure 5-89: n balanced input ports

"Uncoupled" This coupon type selection is available only for balanced ports on the DUT. It is used for deembedding a balanced port when it is desired to treat the single-ended legs of the port as uncoupled entities.

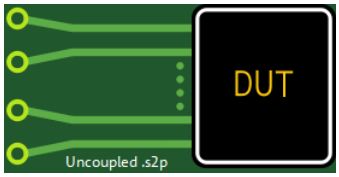


Figure 5-90: n uncoupled balanced input ports

"Coupled" This coupon type is available for DUTs with two or more single-ended or balanced inputs (outputs). It is suitable for a multiport DUT with coupled inputs (outputs) that have to be deembedded as a single unit.
Coupled coupons are drawn with a brace to highlight the difference between coupled and uncoupled ports.

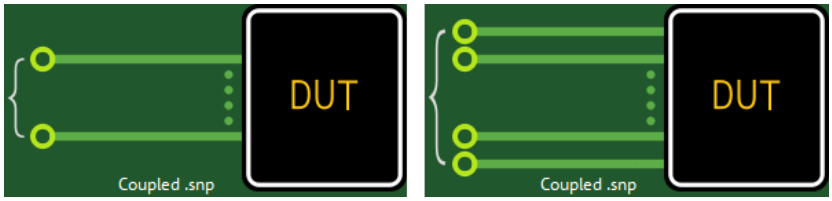


Figure 5-91: n > 1 coupled input ports (single-ended | balanced)

Remote command:
`CALCulate:FMODel:DEASsistant:FIXTure:LEFT:TYPE`
`CALCulate:FMODel:DEASsistant:FIXTure:RIGHt:TYPE`
`CALCulate:FMODel:DEASsistant:PRESet`

Use Same Coupon Left and Right
This control is only active if you selected a two-sided DUT and the same coupon model for the left and right side.

If enabled, then in the measurement phase only the data of a single coupon ("Coupon A") must be measured or loaded from file.

Note: For DeltaCal this is always true and hence this control is hidden.

Remote command:
CALCulate:FMODel:DEASsistant:SAMCoupon
CALCulate:FMODel:DEASsistant:PRESet

Remove Active Deembedding
Use this button to remove the fixture deembedding and to deactivate all other deembedding in the active channel (see"All Deembedding Activated" on page 747).

Note that the "Deembedding Assistant" works slightly different than the legacy Fixture modeling tool panel. In the "Deembedding Assistant", once applied, the deembedding remains active until cleared or (partly) overwritten. The "Fixture Modeling" dialog resets all deembedding files of a specific type (single-ended, balanced, port set), every time you press the "Apply" button.

Overview
This button takes you to the Overview panel of the "Offset Embed" dock widget, which allows you to interact with the individual components of the deembedding calculation chain.

To reopen the "Deembedding Assistant" panel, you have to click/tap on the "Deembed Assistant" tab caption once again.

<Tool> Advanced Settings
Opens the "Advanced Settings" dialog of the selected fixture tool:
- "ISD Advanced Settings" on page 720
- "SFD Advanced Settings" on page 724
- "EZD Advanced Settings" on page 725

Page 2: Measure / Load and Apply

The second page of the "Deembedding Assistant" dock widget allows you to:

- Measure one or more test coupons, or load their S-parameter data from file
- Measure DUT + test fixture
- Send the measured/loaded data to the selected fixture tool and apply the calculated deembedding

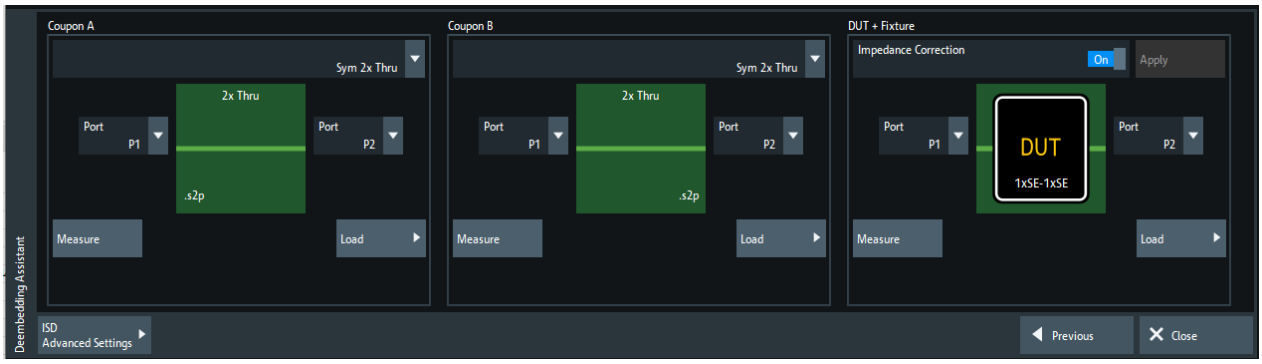


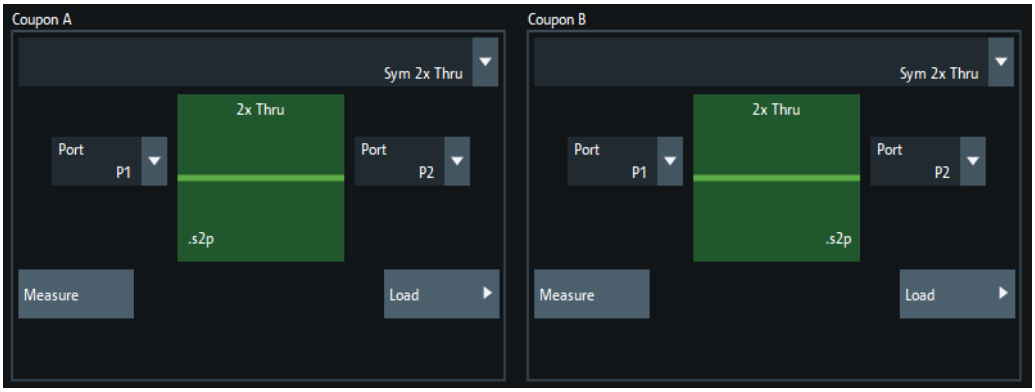
Figure 5-92: Measure / Load and Apply for ISD (SFD, EZD similar)

If you select Use Same Coupon Left and Right on the topology page, then only one set of test coupons can be measured (or loaded from file). The results are used for both sides of the DUT.

With two coupons, "Coupon A" is used to deembed the left-hand side of the DUT and "Coupon B" is used to deembed the right-hand side.

Coupon A / Coupon B section

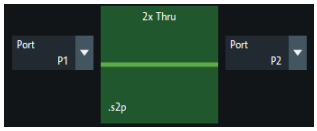
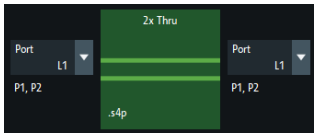
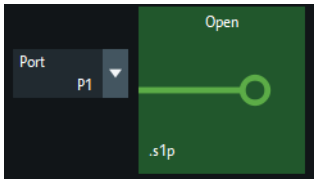
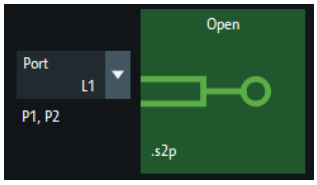
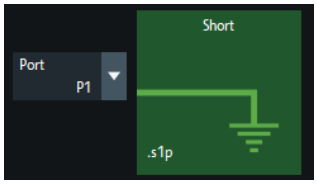
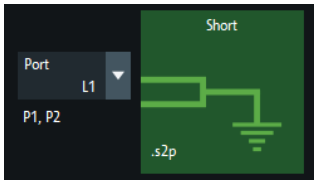
In the "Coupon A" and "Coupon B" section, you can select one or more test coupons and, for each of them, either measure its S-parameters at selected logical ports or load them from a Touchstone file.



Depending on the selected DUT, a different set of test coupon types is supported.

Test coupon	"Sym 2x Thru"	"1x Open"	"1x Short"	"1x Open, 1x Short"
ISD	☑	☑	☑	☑
SFD	☑	☑	☑	☐
EZD	☑	☐	☐	☐

Depending on the selected test coupon type, the number of logical ports and the graphical representation changes:

Test coupon type	Single-ended	Balanced
"Sym 2x Thru"		
"1x Open"		
"1x Short"		

Use the Balanced Ports dialog to create the single-ended and balanced ports that are required for the measurements.

There are several rules to be considered with the coupon measurement interface.

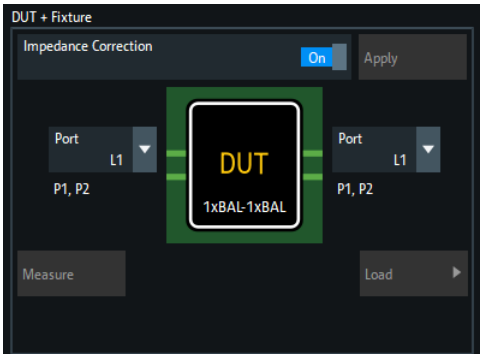
- The port selection drop-down allows you to select from any available port of the correct type on the instrument.
  - For single-ended coupons, any physical port can be selected, even if that port is part of a logical balanced port or port set.
  - For balanced coupons, you can only select a preciously defined logical port.
- The ports used for the measurement must be unique. That is, you cannot select P1 for both sides of a "Sym 2x Thru" coupon. However, you can select the same port for both measurements in a "1x Open, 1x Short" coupon, which requires a connection change before the second measurement.
- The "Load" button allows you to load a Touchstone file instead of measuring the coupon. The graphic displays the Touchstone file's required "cardinality".

Remote command:

```
CALCulate:FMODel:DEASsistant:COUPon:LEFT:TYPE
CALCulate:FMODel:DEASsistant:COUPon:RIGHt:TYPE
CALCulate:FMODel:DEASsistant:COUPon:LEFT:PORT
CALCulate:FMODel:DEASsistant:COUPon:RIGHt:PORT
CALCulate:FMODel:DEASsistant:COUPon:LEFT:MEASure
CALCulate:FMODel:DEASsistant:COUPon:RIGHt:MEASure
CALCulate:FMODel:DEASsistant:COUPon:LEFT:FILE
CALCulate:FMODel:DEASsistant:COUPon:RIGHt:FILE
CALCulate:FMODel:DEASsistant:COUPon:LEFT:CLEar
CALCulate:FMODel:DEASsistant:COUPon:RIGHt:CLEar
```

DUT + Fixture section

After you have measured the coupons or loaded their data from file, you can proceed with the DUT and test fixture assembly. Either measure the assembly, or load its data from file.

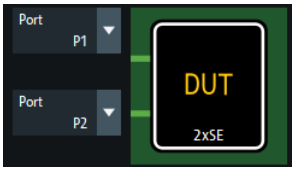


For measuring coupons, choose:

- A single-ended or balanced port for the 1x DUT sides



- Two single-ended ports for the 2x single-ended DUT sides



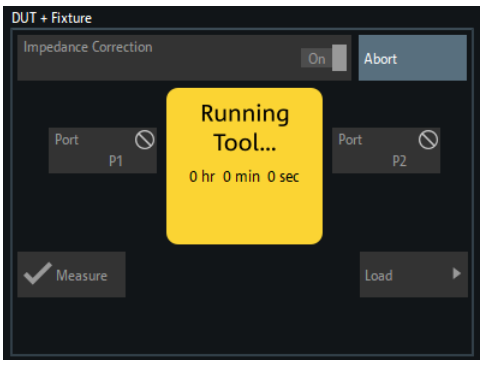
- A range of single-ended or balanced ports for the nx DUT sides



If you want to apply impedance correction, switch "Impedance Correction" ON and "Measure" the DUT with the fixture or "Load" the data from Touchstone file.

For EZD, "Load" is disabled if you want fo select the "Deembedding Plane" manually (see EZD Advanced Settings).

Once the coupons (and the DUT) are measured, the "Apply" button becomes active. Use it to run the tool and, if successful, perform the deembedding on the selected ports.



The deembedding of single-ended and decoupled ports is assigned as Single Ended, the deembedding of balanced ports as Balanced, and the deembedding of coupled as Port Sets deembedding.

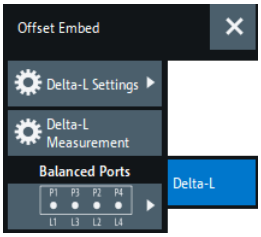
Remote command:

```
CALCulate:FMODel:DEASsistant:IMPCorrect
CALCulate:FMODel:DEASsistant:DUT:PORT
CALCulate:FMODel:DEASsistant:DUT:MEASure
CALCulate:FMODel:DEASsistant:RUN
CALCulate:FMODel:DEASsistant:DUT:FILE
CALCulate:FMODel:DEASsistant:DUT:CLEar
```

5.13.12 Delta-L tab

The controls on the "Delta-L" tab allow you to set up and perform a Delta-L PCB characterization.

The "Delta-L" tab is only visible if option R&S ZNB-K231 is installed on the instrument. See Section 4.7.12, "Delta-L 4.0 PCB characterization", on page 256.



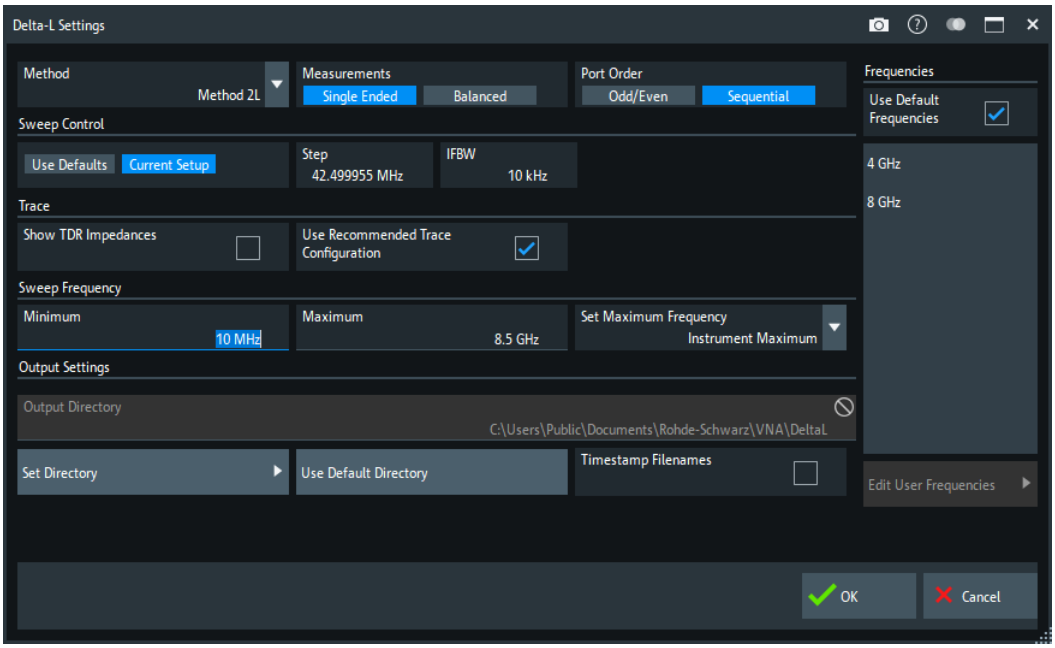
- The "Delta-L Settings..." button opens the Delta-L Settings dialog, which allows you to configure a Delta-L measurement.
- The "Delta-L Measurement" button runs the configured measurement.
- The "Balanced Ports" button opens a dialog that allows you to enable/disable physical ports and to define logical ports (balanced or unbalanced) in the active channel. See "Balanced Ports dialog" on page 296.

If either multiple DUTs are configured (see Section 5.12.2.3, "Define Parallel Measurement dialog", on page 694) or the "Fixture Simulator" is disabled for the related channel (see "Fixture Simulator" on page 672), the "Balanced Ports" button is inactive (grayed out).

5.13.12.1 Delta-L Settings dialog

From the "Detal-L Settings" dialog, you can set up the Delta-L measurement channel and how the external tool is run.

Access: Channel – [Offset Embed] > "Delta-L" > "Delta-L Settings..."



Method
Selects the Delta-L+ characterization method to be used. Default is "Method 2L". For background information, see Delta-L 4.0 PCB characterization.

"Method 1L" handles single-ended and balanced measurement results differently; activate "1L Diff. Mode" for balanced measurements. For "Method 2L" and "Method 3L" measurements can also be single-ended or balanced, but the tool does not distinguish between them.

Remote command:
`CALCulate:FMODel:DELT:METHod`
`CALCulate:FMODel:DELT:M1L:DIFFmode`

Measurements
Defines whether single-ended or balanced ports are measured for Delta-L characterizations.

Remote command:
`CALCulate:FMODel:DELT:MEASurement`

Port Order
Declares how the DUT is connected to the measurement system. Choosing the wrong port ordering for your setup, can result in erroneous results.

"Sequential" Ports 1, 2 and 3, 4 form input/output pairs

"Odd / Even" Ports 1, 3 and 2, 4 form input/output pairs

Remote command:
```
CALCulate:FMODel:DELT:PORDer
```

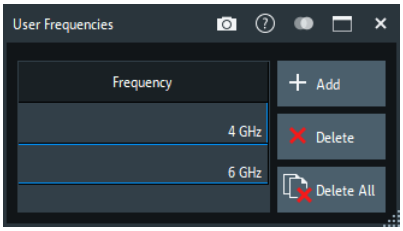
Options
Enables or disables the improved Delta-L algorithms.

Note that the improved Delta-L algorithms are only available with newer versions of the AITT-DL tool that option R&S ZNB-K231 uses under the hood.

Remote command:
```
CALCulate:FMODel:DELT:RESonance
CALCulate:FMODel:DELT:CUToff:STATe
CALCulate:FMODel:DELT:CUToff:FREQuency
CALCulate:FMODel:DELT:FITTing:ORDer
```

Frequencies
Delta-L computes the uncertainty in the loss-per-inch calculation at specified frequencies.

The "Frequencies" table is pre-populated with the frequencies recommended by Intel. If you want to enter your own values, disable "Use Default Frequencies" and select "Edit User Frequencies..." to bring up the "User Frequencies" dialog.



Remote command:
```
CALCulate:FMODel:DELT:FREQuencies:USEDefault
CALCulate:FMODel:DELT:FREQuencies:CURRent?
CALCulate:FMODel:DELT:FREQuencies:CURRent:COUNt?
CALCulate:FMODel:DELT:FREQuencies:DEFault?
CALCulate:FMODel:DELT:FREQuencies:DEFault:COUNt?
CALCulate:FMODel:DELT:FREQuencies:USER?
CALCulate:FMODel:DELT:FREQuencies:USER:COUNt?
CALCulate:FMODel:DELT:FREQuencies:USER:ADD
CALCulate:FMODel:DELT:FREQuencies:USER:DELete
CALCulate:FMODel:DELT:FREQuencies:USER:DELete:ALL
```

Sweep Control
The Delta-L measurement channel can either "Use Defaults" (10 MHz step size and 1 kHz IF bandwidth), or the "Current Sweep" settings (see "Freq Step Size" on page 537, and "Bandwidth" on page 534).

Remote command:
```
CALCulate:FMODel:DELT:SWEep:CONTrol
CALCulate:FMODel:DELT:SWEep:CONTrol:STEP?
CALCulate:FMODel:DELT:SWEep:CONTrol:IFBW?
```

Sweep Frequency
Defines the sweep range of the Delta-L channel. The entered "Minimum" and "Maximum" values are set as start and stop frequency when the Delta-L settings are applied using "OK". Furthermore, the VNA collects data for the LPI computation a the "Minimum" and "Maximum" frequencies (in addition to the other specified "Frequencies" on page 773).

The default "Minimum" is the standard Delta-L start frequency of 10 MHz. The default "Maximum" is either 20 GHz or the instrument's maximum frequency, whichever is smaller. Buttons are available to set "Maximum" to the standard Delta-L stop frequencies of 20 GHz and 40 GHz (if within the instrument's frequency range), and to the instrument's maximum frequency.

If you set the "Minimum" or "Maximum" to a non-standard value, a popup indicates that the Delta-L measurement possibly does not provide accurate results.

Remote command:
```
CALCulate:FMODel:DELT:SWEep:FREQuency:MINimum
CALCulate:FMODel:DELT:SWEep:FREQuency:MAXimum
```

Trace
On OK, the VNA deletes the existing traces and sets up the traces whose measurement is required for the selected Delta-L Method.

If **"Use Recommended Trace Configuration"** is enabled, then each of the new traces is displayed in a separate diagram. Otherwise, the VNA places all traces in a single diagram.

If the Time domain analysis option R&S ZNB-K2 is installed, **"Show TDR Impedances"** adds the TDR-based converted impedance traces to the recommended trace configuration.

Remote command:
```
CALCulate:FMODel:DELT:TCONfig
CALCulate:FMODel:DELT:TDR
```

Output Settings
Allows you to specify the directory where the Delta-L tool stores its data.

Default is `C:\Users\Public\Documents\Rohde-Schwarz\Vna\DeltaL`.

If "Timestamp Filenames" is enabled, the names of subsequently generated "Test Coupon" files are prefixed with the current date and time.

Remote command:
CALCulate:FMODel:DELT:DIRectory
CALCulate:FMODel:DELT:DIRectory:DEFault
CALCulate:FMODel:DIRectory:DEFault
CALCulate:FMODel:REName

OK
"OK" applies the Delta-L settings and starts the Delta-L measurement.

5.13.12.2 Delta-L measurement

When the Delta-L measurement starts, the analyzer firmware prepares the Delta-L measurement channel, populates the diagram area with the required measurement traces, and opens the Delta-L Measurement dock widget.

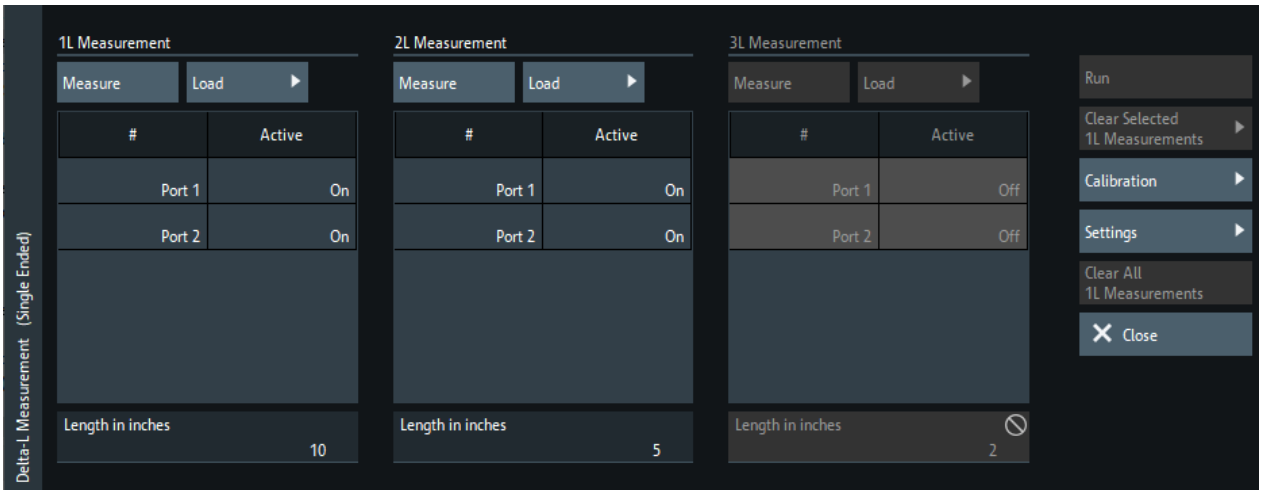
Delta-L Measurement dock widget

The "Delta-L Measurement" dock widget is where coupon measurements are performed and/or corresponding touchstone files are selected.

The Measurements setting in the "Delta-L Settings" dialog determines whether single-ended or balanced can be measured. "No ports available!" in the dock widget and "No Trace" in the diagram area indicates that no ports of the selected type are configured.

Access:

- Channel – [Offset Embed] > "Delta-L" > "Delta-L Measurement"
- Delta-L Settings dialog > "OK"



1L Measurement/2L Measurement/3L Measurement
The settings in the "1L Measurement"/"2L Measurement"/"3L Measurement" sections configure and control the measurement of the 1st/2nd/3rd line.

The "1L Measurement" is always enabled. Depending on the selected "Method" on page 772, the "2L Measurement" and "3L Measurement" can be disabled.

Load File ← 1L Measurement/2L Measurement/3L Measurement
Loads coupon measurement results from a `s2p` or `s4p` Touchstone file.

Remote command:
`CALCulate:FMODel:DELT:M1L:FILename`
`CALCulate:FMODel:DELT:M2L:FILename`
`CALCulate:FMODel:DELT:M3L:FILename`

Measure ← 1L Measurement/2L Measurement/3L Measurement
Measures the full set of S-parameters at the selected ports (full 2- or 4-port).

Remote command:
`CALCulate:FMODel:DELT:M1L:MEASure`
`CALCulate:FMODel:DELT:M2L:MEASure`
`CALCulate:FMODel:DELT:M3L:MEASure`

Ports table ← 1L Measurement/2L Measurement/3L Measurement
Use the controls in the "Active" column to include the respective port into the measurement.

If Method 1L is used, four ports have to be selected in the "1L Measurement" section.

Remote command:
`CALCulate:FMODel:DELT<Ph_pt>:M1L[:STATe]`
`CALCulate:FMODel:DELT<Ph_pt>:M2L[:STATe]`
`CALCulate:FMODel:DELT<Ph_pt>:M3L[:STATe]`

Length ← 1L Measurement/2L Measurement/3L Measurement
Specifies the length of the related line.

Remote command:
`CALCulate:FMODel:DELT:M1L:LENGth`
`CALCulate:FMODel:DELT:M2L:LENGth`
`CALCulate:FMODel:DELT:M3L:LENGth`

Run
Runs the AITT-DL tool with the collected measurement data and the configured Delta-L settings.

The firmware enables this action when sufficient data for the required lines are available (either loaded or measured).

If the AITT-DL tool experiences errors, the analyzer firmware displays additional information (see <Deembedding Tool> Errors dialog).

Remote command:
`CALCulate:FMODel:DELT:RUN`

Clear All 1L Measurements/Clear Selected 1L Measurements
If Method 1L is selected in the "Delta-L Settings" dialog, the VNA firmware builds up a cache of coupon measurements (Touchstone files) in the `DeltaL\1LMeasurementCache` subfolder of the user data folder `C:\Users\Public\Documents\Rohde-Schwarz\Vna`.

"Clear All 1L Measurements" deletes this cache, "Clear Selected 1L Measurements" displays a "Delete files" dialog that allows you to select the cache files to be deleted.

Remote command:

```
CALCulate:FMODel:DELT:M1L:CACHe:CLEar:ALL
CALCulate:FMODel:DELT:M1L:CACHe:CLEar:SELected
```

Delta-L diagram

Upon successfully running the external AITT-DL tool, the VNA displays the "Delta-L" diagram.

2L and 3L

For "Method 2L" and "Method 3L", the "Delta-L" diagram includes two traces for every combination of lines/lengths (12 for "Method 2L"; 12, 13, and 23 for "Method 2L"):

- Raw traces "DeltaL12", ...
- Smoothed traces "DeltaL12S", ...
  Smoothing is performed by the Delta-L algorithm.



Figure 5-93: Method 2L results

Markers are added to the smoothed traces at the analysis frequencies. The marker name indicates to which analysis pair it belongs and the computed Delta-L uncertainty.

Method 1L

For "Method 1L", the Delta-L diagram displays the following traces:

- An insertion loss trace "L<i>" for each touchstone file `L<i>.s<2|4>p` (loaded or measured) in the measurement cache
- An additional trace "StdDev" representing the standard deviation of the 1L analysis at the analysis frequencies



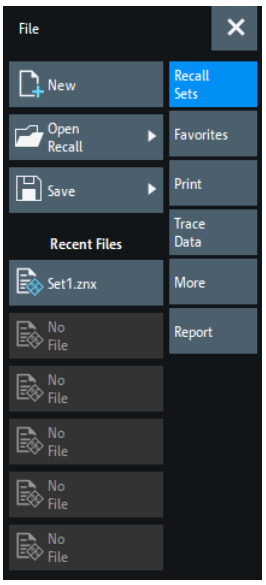
Markers "DeltaL1..." are added to the"StdDev" trace at the analysis frequencies. The marker names indicate the computed Delta-L uncertainties.

5.14 File softtool

The "File" softtool allows you to work with recall sets and trace data.

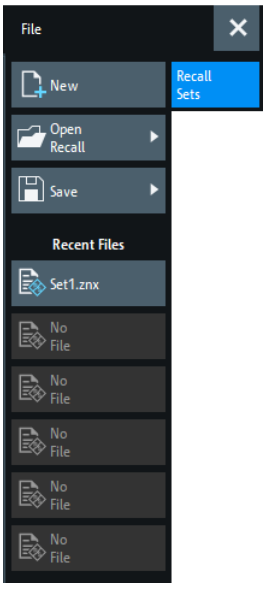
Access:

- System – [File Print], for 2nd generation R&S ZNB models only (order nos. 1334.3330.xx)
- System – [File], for other R&S ZNB and R&S ZNBT



5.14.1 Recall sets tab

A recall set comprises a set of diagrams together with the underlying system, channel, trace and display settings. It can be stored to a VNA recall set file (`*.znx|*.znxml`).



For background information, see Section 4.1.2, "Recall sets", on page 80.

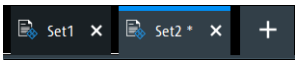
5.14.1.1 Basic Recall Set functions

To create a recall set based on the current analyzer configuration, select Save. To open an existing recall set, select Open Recall.... To create an additional setup, select New.

The R&S ZNB supports two recall set file formats:

- `znx` (default)
  Binary file format, supported by all firmware versions of the R&S ZNB
- `znxml`
  XML based file format, introduced with the R&S ZNA; supported with firmware versions 3.00 a higher

New
Adds a new setup. The default names for new setups are "Set1", "Set2" etc. Recall sets are accessible via tabs in the diagram area:



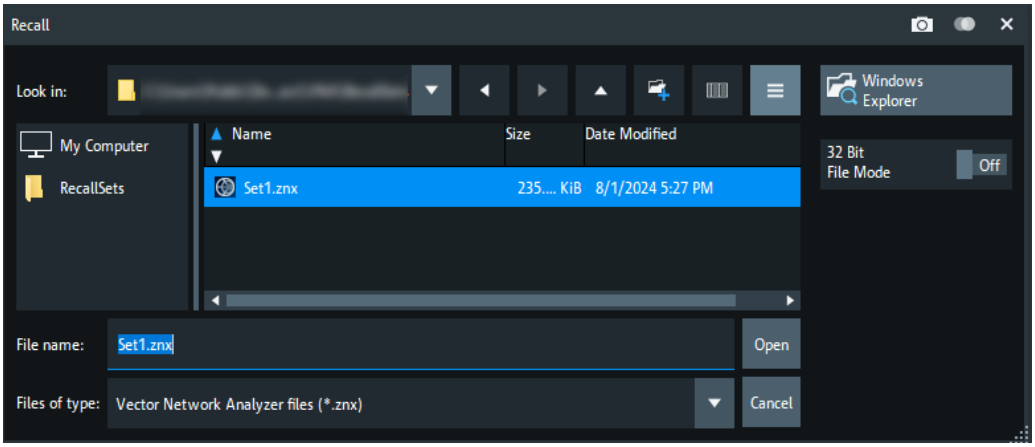
Tip: To open an existing recall set, use "Open Recall...". To rename a setup, use "Save...".

Remote command:
`MEMory:DEFine`

Open Recall...
Loads a recall set from a file (`*.znx|*.znxml`).

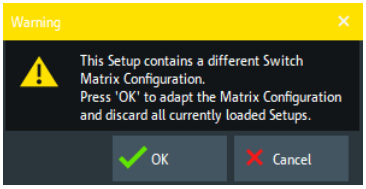
Opens the Open Dialog with suitable file filters.

Enable the "32 Bit File Mode" option on the right-hand side of this dialog if you want to open a `znx` file that was saved by a 32-bit firmware.



If the current setup includes External switch matrices and the recall set to be loaded does not contain a switch matrix configuration, recalling the setup is aborted. You are requested to delete the configured RF connections (see Section 5.17.6.4, "Switch Matrix RF Connections dialog", on page 859) and to remove the switch matrix from the RF paths.

If the recall set to be loaded contains a switch matrix configuration and no differences to the current switch matrix configuration are detected, the recall set is loaded without further inquiry. Otherwise you are asked if you want to adapt the configuration:



Select "Cancel" to abort the recall process or "OK" to open the Restore Switch Matrix Configuration dialog that guides you through the required configuration changes.

Note: In case the recall set could only be loaded after adjusting some incompatible settings, the info message **"The setup data changed while loading"** is displayed. Incompatibilities can arise, for example, if you attempt to load a recall set of a different R&S ZNB model or with different optioning. Not all incompatibilities can be adjusted.

Remote command:
`MMEMory:LOAD:STATe`

Save
Saves the active recall set to a file (`*.znx|*.znxml`).

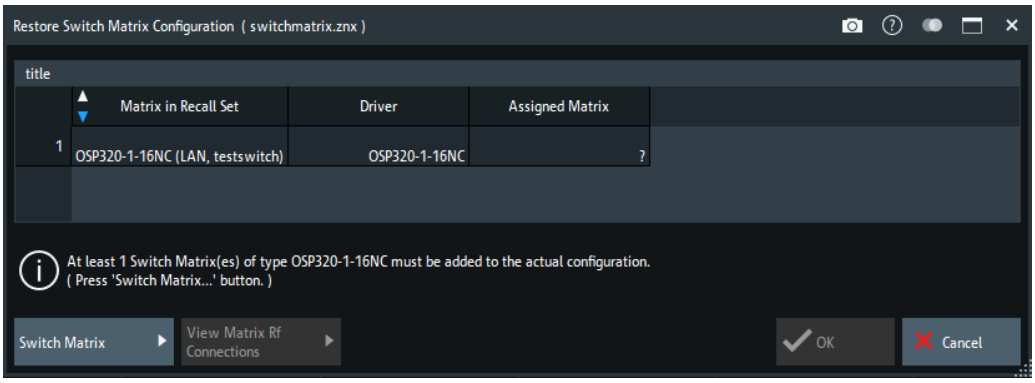
Displays the Save Dialog that allows you to specify a filename and location for the recall set file.

Remote command:
`MMEMory:STORe:STATe`

Recent Files
These buttons represent the recall set history of the current and previous sessions. They open the corresponding recall set.

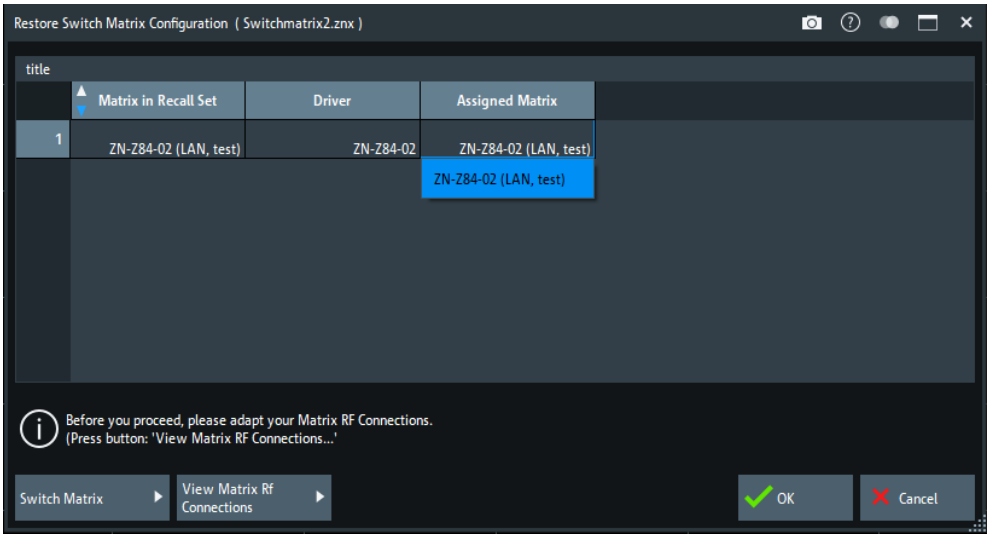
5.14.1.2 Restore Switch Matrix Configuration dialog

When opening an existing recall set file, it can be necessary to adapt the current system configuration to the switch matrix configuration of the recall set. The "Restore Switch Matrix Configuration" dialog guides you through this process.



In the dialog, proceed as follows:

1. **Reassign Matrices:** For each "Matrix in Recall Set", select a switch matrix of the same (driver) type as "Assigned Matrix".



If no switch matrix of the same type exists, select "Switch Matrix" to open the External Matrices dialog that allows you to register/configure additional switch matrices.

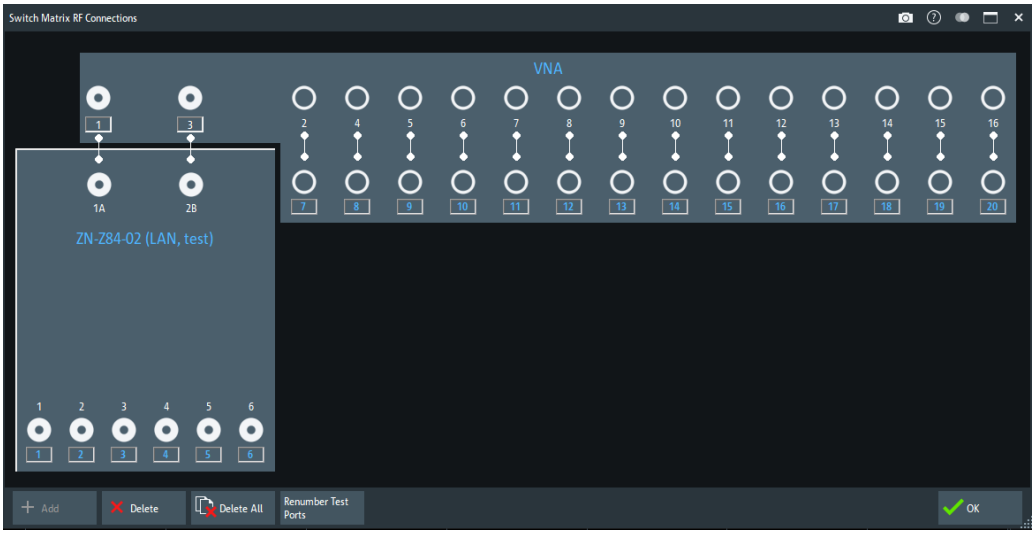
2. **Restore RF connections:** The recall set's switch matrix configuration comprises the physical RF connections of the related switch matrices. For each "Matrix in Recall Set", select the corresponding row and select "View Matrix RF Connections..." to view the respective matrix RF connections. Make sure to connect the "Assigned Matrix" accordingly.

Background information

Refer to Section 4.7.30, "External switch matrices", on page 271.

View Matrix RF Connections...

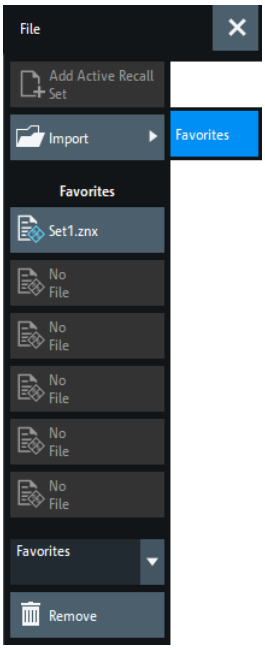
This dialog presents the RF connections of a selected switch matrix as specified in the recall set you are about to load.



It is a read-only version of the Switch Matrix RF Connections dialog you can later use to modify the actual configuration.

5.14.2 Favorites tab

The "Favorites" tab allows you to manage a list of favorite recall sets.



A favorite is actually a path to the related recall set, i.e.:

- If the recall set is modified, then the modified recall set is loaded the next time the favorite is selected.
- If the recall set is moved or deleted, the corresponding favorite is broken.

The firmware can manage up to 6 favorites. New favorites are always created at the topmost spare position in the favorites list. If the list is complete (i.e. if there are no more spare positions), new favorites can only be added after existing ones were removed.

Add Active Recall Set
Adds the active recall set to the list of favorites.

This button is only active if the current setup was loaded from or saved to a recall set.

Import
Opens a file browser that allows you to add an arbitrary recall set to the favorites list.

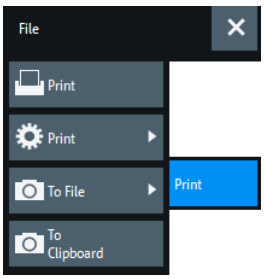
Remote command:
`MMEMory:FAVorite<FavId>`

Favorites section
Select one of the active buttons to open the corresponding recall set.

Favorites combo-box / Remove
Use the "Favorites" combo-box to select the favorite to be removed from the list, then select "Remove".

5.14.3 Print tab

The buttons on the "Print" tab allow you to send the diagrams of the active setup to an external printer, to a file or to the clipboard. Content and layout can be defined in the Printer Setup dialog.



Print
Prints the diagrams using the current content, printer and page settings (see Section 5.14.4, "Printer Setup dialog", on page 785).

Remote command:
The `HCOPy...` commands provide the printer settings; see HCOPy commands.
`HCOPy[:IMMediate]` initiates printing.

Print...
Opens a dialog that allows to define the content, printer and page settings (see Section 5.14.4, "Printer Setup dialog", on page 785).

To File...
Opens the "Save Image" dialog that allows you to select a graphics file type (`*.bmp, *.jpg | *.jpeg, *.pdf, *.png, *.svg`) and to save diagram content to a file of this type (see also "Save Dialog" on page 106).

Note that the diagram content to be saved can be configured in the Content tab > "Print Charts" group of the "Printer Setup" dialog. E.g., if you select "All diagrams on their own page" there, then only the content of the active diagram is saved to file. If you select "No diagram at all", then "To File..." is disabled.

Remote command:
`HCOPy:DESTination`

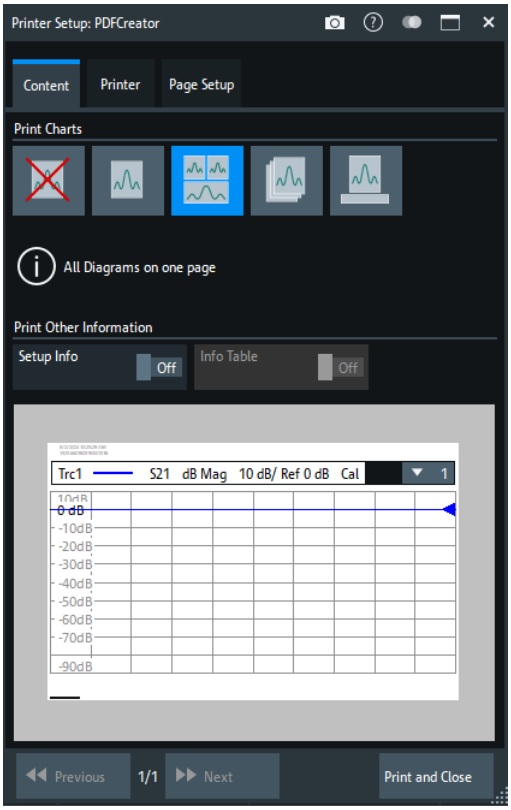
To Clipboard
Copies diagram content to the clipboard, from where you can paste it into another application.

Note that the diagram content to be copied can be configured in the Content tab > "Print Charts" group of the "Printer Setup" dialog. E.g., if you select "All diagrams on their own page" there, then only the content of the active diagram is copied to clipboard. If you select "No diagram at all", then "To Clipboard" is disabled.

5.14.4 Printer Setup dialog

The "Printer Setup" dialog specifies how the diagram content of, and/or other information on the active recall set is printed. Printer options are specified in three tabs. The lower part of the dialog shows a preview of the print.

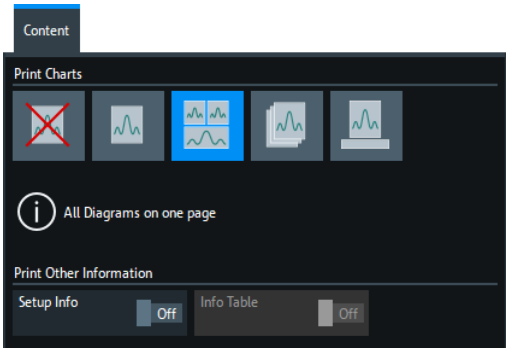
Access: System – [File] > "Print" > "Print"



The `HCOPy...` commands provide the printer settings; see Section 7.3.7, "HCOPy commands", on page 1242.

5.14.4.1 Content tab

The "Content" tab allows you to select the diagram content to be printed.



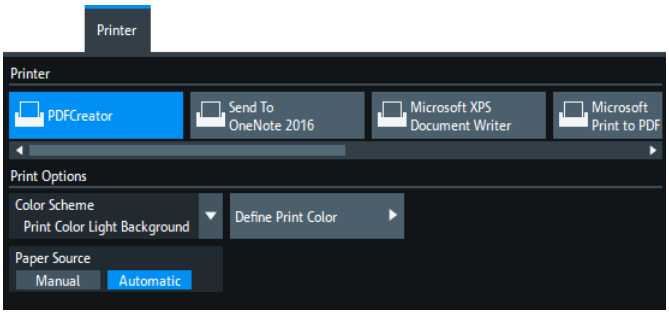
The selected item in the " Print Charts" group specifies how the diagram area of the active recall set is printed. Currently the following options are offered: "No Diagram at all", "Active Diagram only", "All diagrams on one page", "All diagrams on their own page" and "Hard copy of diagram area". The latter is a screenshot of the diagram area, which preserves colors and layout.

Note that the " Print Charts" setting also affects the To File... and To Clipboard actions of the Print tab.

If selected in the "Print other information" group, the content of the "Info Table" (see "Info Table: Show / Position" on page 803) and/or "Setup Info" (see "Setup tab" on page 828) is printed on additional pages.

5.14.4.2 Printer tab

The "Printer" settings select one of the installed printers and specify printer options.



Printers can be installed using the Windows® "Devices and Printers" functionality; see also Section 3.1.11.4, "Connecting a printer", on page 30.

Print Options
Allows you to select and/or define the color scheme for printing.

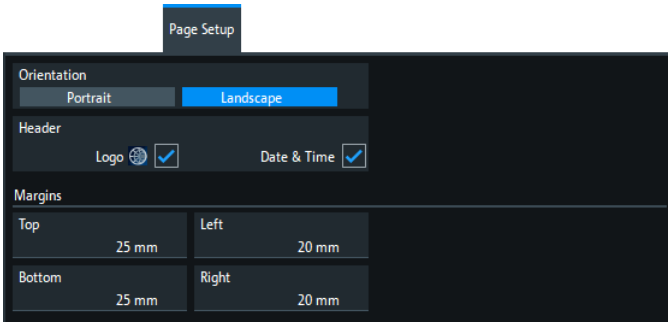
The "Define Print Color..." button opens the Define User Color Scheme dialog which allows you to create or modify your own print color scheme.

The R&S ZNB now uses different "User Defined" color schemes for display and printing.

Remote command:
`HCOPy:PAGE:COLor`

5.14.4.3 Page Setup tab

The "Page Setup" settings are visualized in the preview page in the lower part of the dialog.



Tip: The printer settings are not affected by a preset of the R&S ZNB. Use the "Remote" tab in the System Config dialog to restore default settings.

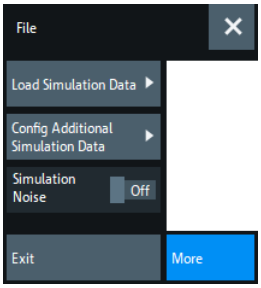
5.14.5 Trace Data tab

See Section 5.5.8, "Trace Data tab", on page 475.

5.14.6 More tab

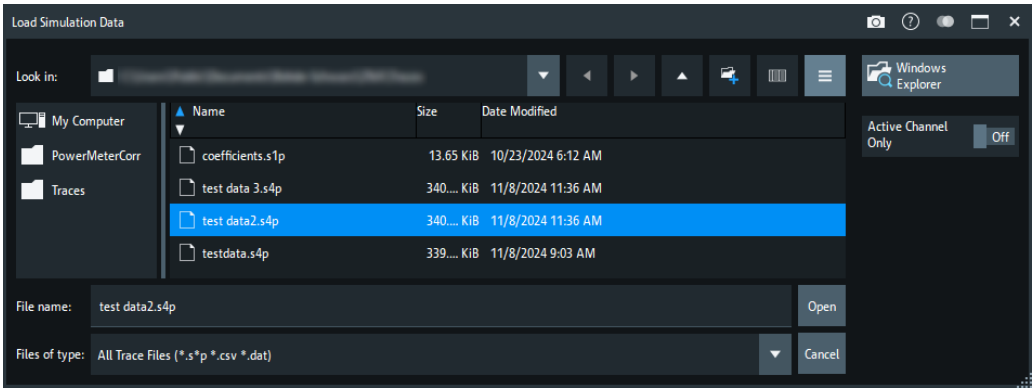
The buttons on the "More" tab allow you to load simulation data or close the VNA application.

Loading simulation data and adding simulation noise is only available for the PC firmware simulation (see Section 4.7.32, "R&S ZNXSIM", on page 284).



Load Simulation Data...
Imports simulation data into the current setup (firmware simulation only; see Simulation data).

The analyzer opens the "Load Simulation Data" dialog that lets you select a trace file (`*.sNp, *.csv, *.dat`). As an option, you can load the simulation data into the "Active Channel Only", allowing you to analyze different simulation data in different channels of a setup.



Tip: Using the standard trace file export functionality, the loaded data can be converted to different formats.

Remote command:
`MMEMory:SIMulation:LOAD`

Config Additional Simulation Data...
Allows you to import port-specific simulation data into the current channel (firmware simulation only; see Simulation data).

Opens the Configure Additional Simulation Data dialog.

Simulation Noise
Allows you to add random noise to the loaded simulation data to let the traces appear more "realistic" (see Section 4.7.32.2, "Simulation noise", on page 287). By default, "Simulation Noise" is inactive.

Make sure to deactivate "Simulation Noise" before exporting traces to file. Otherwise your exported results comprise an "artificial" random component.

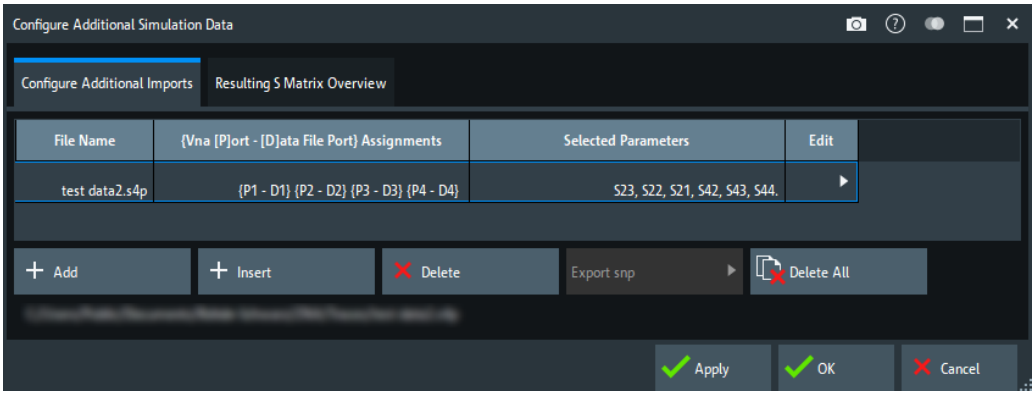
Exit
Persists the current state of all opened setups and ends the analyzer session. These states are automatically recalled when the analyzer application is restarted.

Tip: This button is equivalent to the Windows® "Close window" command and to the close icon in the title bar of the main firmware application window.

5.14.7 Configure Additional Simulation Data dialog

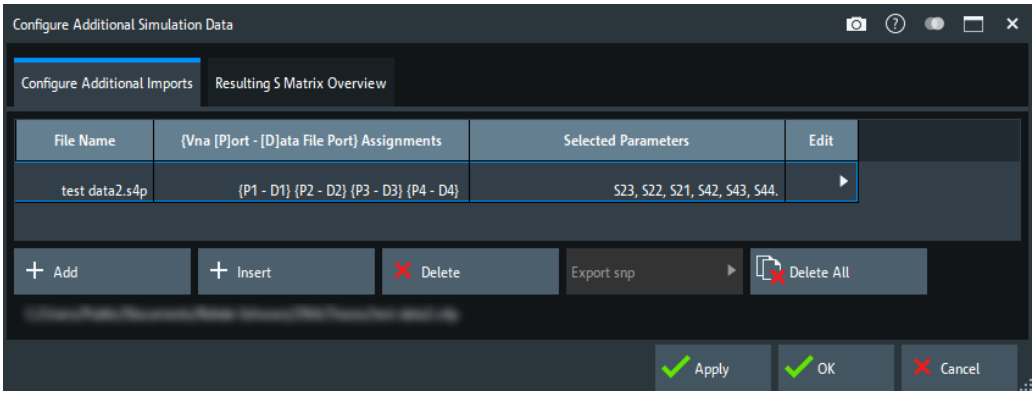
The "Configure Additional Simulation Data" allows you to define port-specific S-parameter data for the current channel (see Section 4.7.32.1, "Simulation data", on page 286).

Access: System – [File] > "More" > "Config Additional Simulation Data..."



5.14.7.1 Configure Additional Imports tab

On the "Configure Additional Imports" tab, you can load additional Touchstone files and overwrite the current channel's S-parameter data by selected S-parameter traces from these files. These data take precedence over global, setup-specific, and channel-specific data (see "Load Simulation Data..." on page 787).



The source data files (first column), the mapping between physical ports and data file ports (second column), and the S-parameters to be overwritten (third column) can be defined during Add/Insert. The text box below the table shows the full path to the data file of the selected row.

Precedence of S-parameter imports

If the same VNA S-parameter is overwritten in several rows, the highest positioned row takes precedence. The resulting overwrites of the channel-specific/setup-specific/ global simulation data are summarized in the Resulting S Matrix Overview tab.

Performance aspects

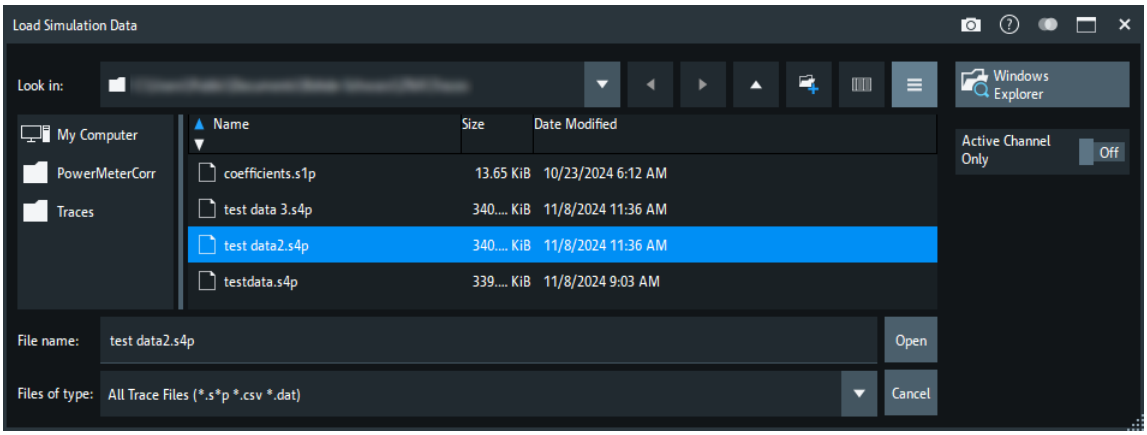
Loading additional simulation data from many (> 5) different files can slow down the firmware significantly. To keep the number of files as low as possible, the Export snp function allows you to export the resulting VNA S-matrix into a single Touchstone file. You can then delete all additional inputs and load this file as channel- or setup-specific simulation data instead (see "Load Simulation Data..." on page 787).

Add/Insert
Allows you to import S-parameter traces from additional Touchstone files into the active channel's simulation data (globally defined, or set using Load Simulation Data...).

"Add" puts them at the end of the import definition table, "Insert" at the position below the currently selected row. Note that the order of rows is significant (see "Precedence of S-parameter imports" on page 789).

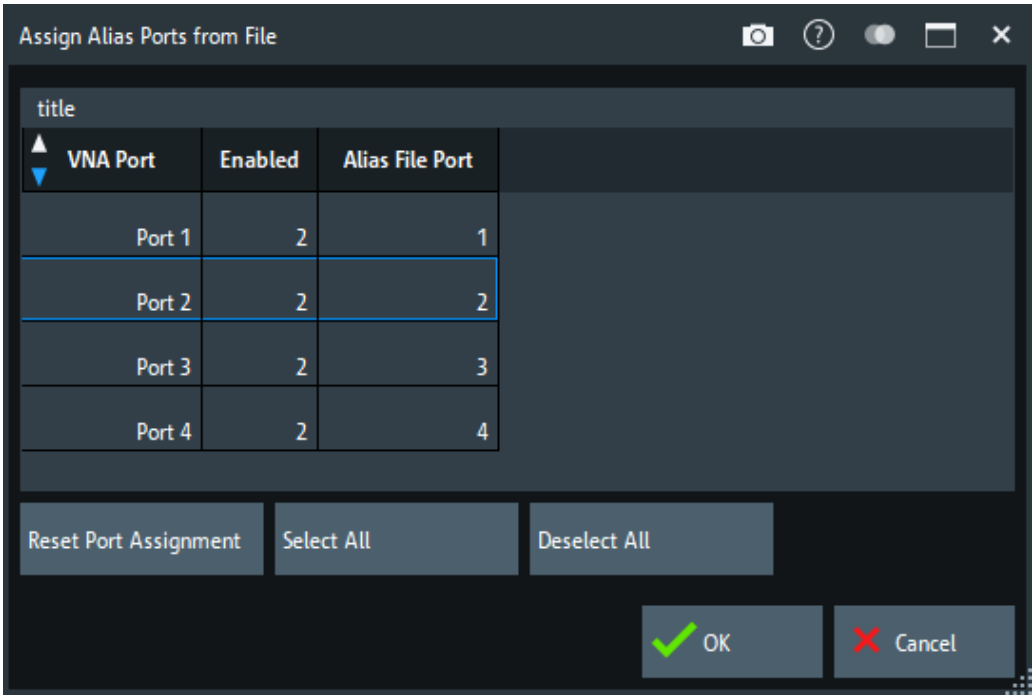
In both cases, the import proceeds in three steps.

Step 1: Load simulation data ← Add/Insert
The "Load Simulation Data" dialog, which is opened on "Add"/"Insert", allows you to select the Touchstone file from which you want to import additional simulation data.



By selecting "Open", you proceed to Step 2: Assign alias ports from file, "Cancel" discards the import definition.

Step 2: Assign alias ports from file ← Add/Insert
The "Assign Alias Ports from File" dialog allows you to select the physical VNA ports whose simulated S-parameter data you want to overwrite ("Enabled"), and their related ports in the selected data file ("Alias File Port").



With "Ok" you proceed to the final Step 3: Select S-parameters, "Cancel" discards the import definition.

Step 3: Select S-parameters ← Add/Insert
In the final step, you select the S-parameter data you actually want to overwrite. The GUI is described in Section 5.5.8.3, "Select Parameter dialog", on page 482. The port numbers refer to the physical VNA ports (not the data file ports).



Select at least one trace to enable the "Ok" button. Selecting "Ok" applies, selecting "Cancel" discards your import definition.

Remote command:
`MMEMory:SIMulation:SDATa<Idx>`
`MMEMory:SIMulation:SDATa<Idx>:LOAD`

Delete/Delete All
Deletes the selected or all additional imports.

Remote command:
`MMEMory:SIMulation:SDATa<Idx>:DELete`
`MMEMory:SIMulation:DELete:ALL`

Export snp
Allows you to export the active channel's complete VNA S-matrix into a single Touchstone file.

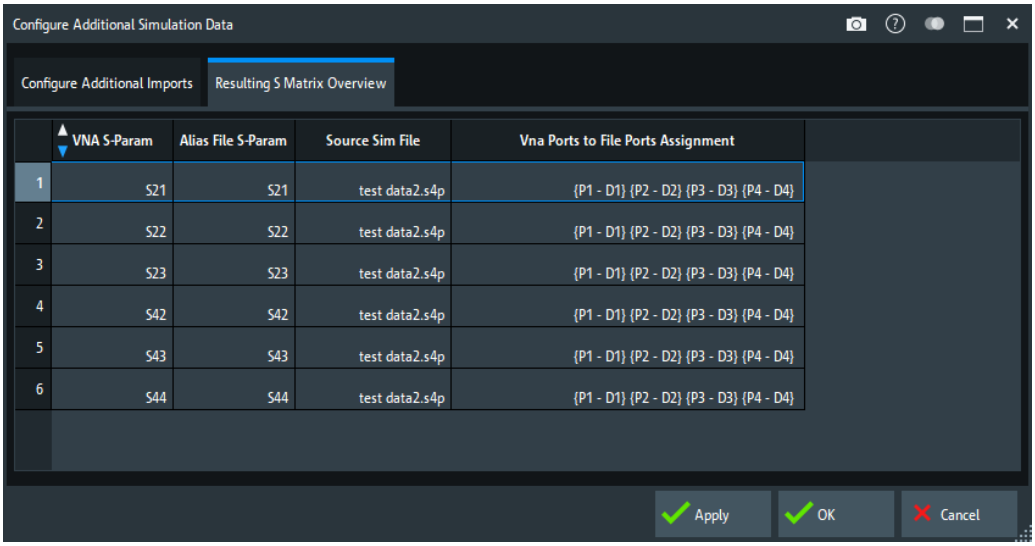
The S-parameters defined by additional imports are summarized on the Resulting S Matrix Overview tab. The remaining ones are taken from the channel-specific, setup-specific, or global simulation data.

For a possible application, see "Performance aspects" on page 789.

5.14.7.2 Resulting S Matrix Overview tab

The "Resulting S Matrix Overview" tab summarizes the S-parameters defined by additional imports. See Section 5.14.7.1, "Configure Additional Imports tab", on page 789.

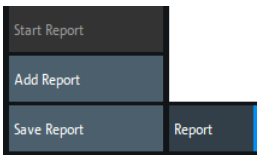
The remaining S-parameters that are required for the channel simulation are taken from the channel-specific, setup-specific, or global simulation data (see "Load Simulation Data..." on page 787).



5.14.8 Report tab

The buttons on the "Report" tab allow you to generate a PDF report comprising:

- Setup information
- The diagrams of the active setup and the content of the info table, if visible (optional).
  The diagrams can be printed in different formats and with configurable header text.



Start Report
Initializes a report by collecting setup information.

You can immediately go to Save Report and save only the setup information as PDF file, or use Add Report to add chart hardcopies (in different formats)

Remote command:
`SYSTem:REPort:STARt`

Add Report
Opens the "Chart Selection" dialog that allows you to add the diagrams of the active recall set to the report. Use the "Add" button to save the defined settings for the report.

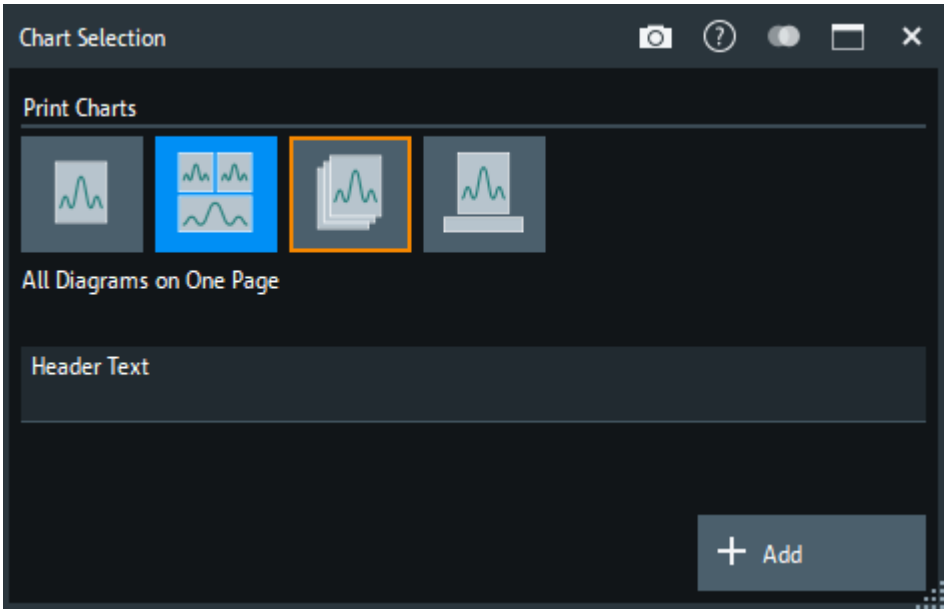


Figure 5-94: Chart Selection dialog

The selected item in the "Print Charts" group specifies the print layout. Currently the following options are offered: "Active Diagram only", "All diagrams on one page", "All diagrams on their own page" and "Hard copy of diagram area". The latter is a screen-shot of the diagram area, which preserves colors and layout.

Adding a "Header Text" to the diagram pages is optional.

Note that you can use "Add Report" multiple times to print the diagrams in different formats.

Remote command:
`SYSTem:REPort:ADD`

Save Report
Allows you to save the contents to a PDF file.

Remote command:
`SYSTem:REPort:SAVE`

5.15 Tools softtool

The "Tools" softtool gives access to applications and tools that extend the functionality of the analyzer firmware.

Access: System – [Applic]

5.15.1 External Tools tab

The "External Tools" tab gives access to pre-installed and user-defined external tools.

Access: System – [Applic] > "External Tools"



GPIB Explorer
Opens a tool that allows you to connect to the analyzer, obtain an overview of all implemented remote control programs, test programs, compile and run test scripts. For a detailed description, refer to Section 6.1.3, "GPIB Explorer", on page 870.

Protocol Wordpad
Convenience function for opening the WordPad word processor, e.g. for creating ad hoc measurement protocols, reports etc.

Tool 3 ... Tool 8
Allows you to add your own external tools. Any new shortcut in the `C:\Users\Public\Documents\Rohde-Schwarz\Vna\External Tools` directory replaces one of the buttons.

Title and Bar Task Bar On
Displays or hides the title bar and the taskbar across the bottom of the screen. Typically you can use the taskbar to change between the VNA application and other external tools. See also Section 3.3.2.1, "Title bar", on page 49.

Screen Keyboard
Opens the Windows "On-Screen Keyboard". This tool allows you to enter characters, in particular letters, if an input field cannot call up the analyzer's own on-screen keyboard, and if no external keyboard is connected.

See also Section 3.3.6.4, "Using the Windows® on-screen keyboard", on page 64.

Windows Explorer
Opens the Windows Explorer and shows you the contents of the `C:\Users\Public\Documents\Rohde-Schwarz\Vna\External Tools` application shortcut directory.

5.16 Display softtool

The "Display" softtool provides all display settings and the functions for activating, modifying and arranging different diagrams.

Access: System – [Display]

Related information
Refer to the following sections:

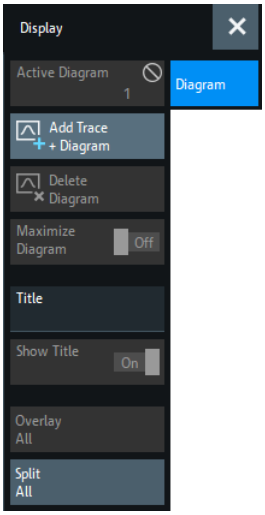
- Section 4.1.3, "Traces, channels and diagrams", on page 80
- Section 4.2.1, "Display elements of a diagram", on page 93
- Section 3.3.5, "Handling diagrams, traces, and markers", on page 57

5.16.1 Diagram tab

Selects a diagram as the active diagram, defines a title, deletes or adds diagrams and arranges them on the screen. Many of the functions are unavailable if the active recall set contains only one diagram.

Related settings

Use the icons in the toolbar to add diagrams and traces. Use the "Zoom Active Trc" icon to zoom into a rectangular portion inside a diagram. See also Section 3.3.5, "Handling diagrams, traces, and markers", on page 57 and Section 3.3.7.1, "Using the graphical zoom", on page 65.



Active Diagram
Selects the active diagram.

Each recall set screen can display several diagrams simultaneously, each with a variable number of traces. One of these diagrams and traces is active at each time. The diagram number (or name) in the upper right corner of the active diagram is highlighted. At the same time, the active trace is highlighted in the trace list on top of the active diagram ("Trc3" in the figure below):



The analyzer provides several tools for activating diagrams:

- tap on a point in the diagram to activate the diagram including the last active trace in the diagram.
- tap on a trace list to activate the trace including the corresponding diagram.
- Some of the functions of the Traces tab activate a particular trace including the corresponding diagram.

Remote command:
`DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:CATalog?`

Add Tr+Diag
Creates a diagram and a trace which is displayed in the new diagram. The trace is created with the channel settings of the previous active trace but with default trace settings. The new diagram area is numbered <n>, where <n> is the largest of all existing diagram area numbers plus one.

Tip: Dragging the "Trc+" toolbar icon to a new diagram area is similar to "Add Tr +Diag".

Remote command:
DISPlay[:WINDow<Wnd>][:STATe] ON

Delete Diagram
Deletes the current diagram area including all traces displayed in the diagram area. The remaining diagrams are renumbered; each recall set always contains diagrams with contiguous numbers. "Delete Diag Area" is disabled if the recall set contains only one diagram area: In manual control, each recall set must contain at least one diagram area with one channel and one trace.

Tip: To restore a diagram area that was unintentionally deleted, use the undo functionality.

Remote command:
DISPlay[:WINDow<Wnd>][:STATe] OFF

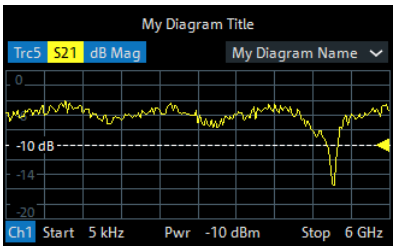
Maximize Diagram
Maximizes the active diagram or restores the previous diagram arrangement.

For other split types, use the functions in the Split tab.

Remote command:
DISPlay[:WINDow<Wnd>]:MAXimize

Title
Defines a title for the Active Diagram.

The visibility of the title area can be toggled using Show Title.



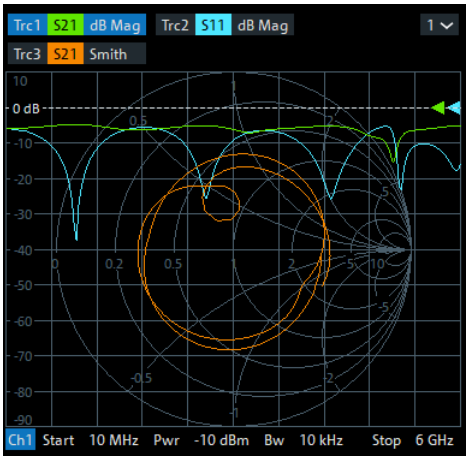
Remote command:
DISPlay[:WINDow<Wnd>]:TITLe:DATA
Via remote control, it is also possible to define a diagram **name**, and to retrieve the lists of diagrams together with their names:
DISPlay[:WINDow<Wnd>]:NAME
DISPlay[:WINDow<Wnd>]:CATalog?

Show Title
Displays or hides the title area of the active diagram.

This property can only be set if Title is non-empty. If "Title" is empty, the title area is always hidden.

Remote command:
DISPlay[:WINDow<Wnd>]:TITLe[:STATe]

Overlay All
Places all traces in a single diagram area which is maximized to occupy the whole screen. This function is available irrespective of the trace format and the channel settings; it is even possible to overlay Cartesian and complex diagrams.

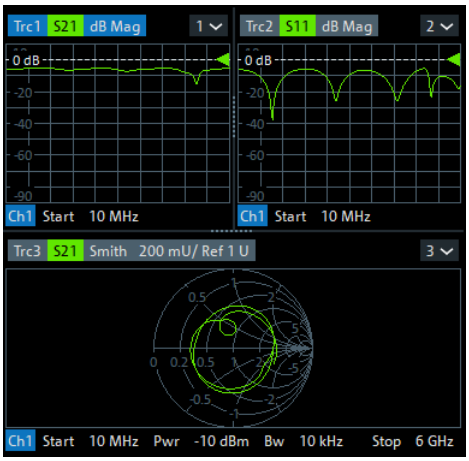


The active trace and active channel is highlighted. The scaling of the axes corresponds to the active trace.

Tip: To hide all traces except one, activate the context menu of the respective trace name segment in the trace list and select "Hide all other Traces".

Remote command:
`DISPlay:LAYout:OVERlay`

Split All
Creates a separate diagram for each trace in the active recall set and automatically arranges those diagrams in the diagram area. Existing diagrams are deleted during this process.

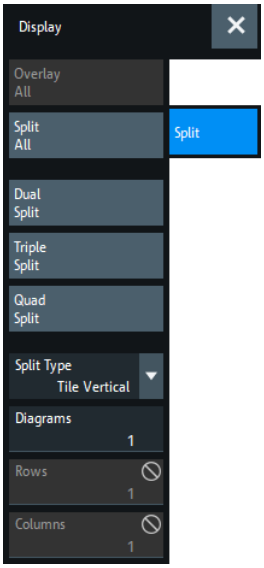


Tip: To vary the size and position of the diagram areas, drag and drop the separating frames or use the functions in the "Split" tab.

Remote command:
`DISPlay:LAYout:SPLit`

5.16.2 Split tab

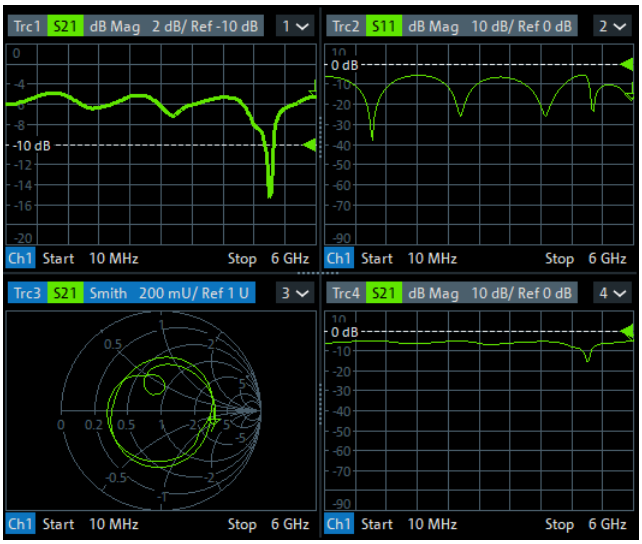
Arranges multiple diagrams on the screen.



Some of the "Split" settings are also available in the Diagram tab. Refer to the following sections:

- "Overlay All" on page 798
- "Split All" on page 798

Dual Split / Triple Split / Quad Split
Splits the diagram area into two (three / four) diagrams and distributes the traces among the diagrams. Traces with different format and channel settings (e.g. Cartesian and complex diagrams) are separated, if possible. Example of four traces in "Quad Split" display:



If less than two (three / four) traces are available, the new diagrams are created with a default trace. Dual (triple / quad) split corresponds to "Split Type: Tile Horizontal" with 2 (3 / 4) diagrams.

Tip: To vary the size and position of the diagrams, drag and drop the separating frames or use the functions in the "Diagram" tab.

Remote command:
No command; display configuration only.

Split Type
The R&S ZNB provides the following split types:

- "Lineup": The diagrams are arranged side by side; each diagram occupies the entire screen height.
- "Stack": The diagrams are arranged one below the other; each diagram occupies the entire screen width.
- "Tile Horizontal": The diagrams are arranged in rows. With 2 (3 / 4) diagrams, the result is equivalent to Dual Split (Triple Split / Quad Split); see "Dual Split / Triple Split / Quad Split" on page 799.
- "Tile Vertical": The diagrams are arranged in columns.
- "Rows + Cols": The diagrams are arranged as a rectangular matrix. The number of rows and columns is as defined in the corresponding input fields.

If the selected number of "Diagrams" exceeds the number of traces, some of the new diagrams are created with a default trace.

Tip: To vary the size and position of the diagrams, drag and drop the separating frames or use the functions in the "Diagram" tab.

Remote command:
`DISPlay:LAYout`

Diagrams / Rows / Columns
Selects the number of "Diagrams" (or "Rows" and "Columns") to which the traces in the active recall set are split. The split is performed according to the selected Split Type.

If the entered number of "Diagrams" exceeds the number of previously existing traces, some of the new diagrams are created with default traces.

For a "Split Type" other than "Rows + Cols", only the total number of "Diagrams" can be specified.

Remote command:
`DISPlay:LAYout:GRID`

Additional Functionality: SCPI Commands
The analyzer provides remote control commands for efficient diagram handling. The commands listed below extend the functionality of the "Display > Diagram" and "Display > Split" softtool panels. For programming examples, refer to Section 8.2.2.6, "Creating diagrams", on page 1664.

Remote command:
`DISPlay:LAYout:APPLy`
`DISPlay:LAYout:DEFine`
`DISPlay:LAYout:EXECute`
`DISPlay:LAYout:JOIN`

5.16.3 Config tab

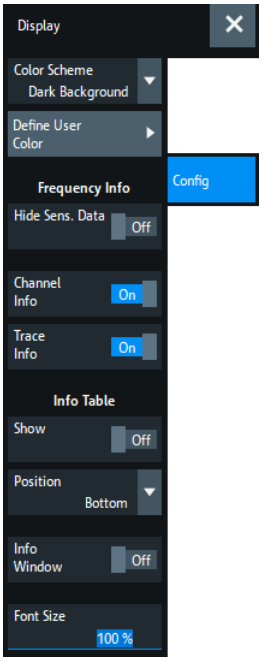
Displays or hides controls and information elements of the screen and controls the appearance of the individual diagrams.

Hiding the controls and information elements leaves more space for the diagrams. All elements can be shown or hidden simultaneously.

Related information

Refer to Section 4.2.1, "Display elements of a diagram", on page 93.

5.16.3.1 Controls on the Config tab



Color Scheme
Controls the colors in the diagram areas. Color schemes are global settings and apply to all active recall sets.

The following predefined color schemes are optimized for the analyzer screen and for color hardcopies, respectively:

- "Dark Background" sets a black background color. The traces and information elements in the diagram areas are displayed in different colors. This setting is usually suitable for observing results on the analyzer screen.
- "Light Background" sets a light background color. The traces and information elements in the diagram areas are displayed in different colors. This setting is suitable for generating color hardcopies of the screen.

The following predefined color schemes can be appropriate for generating black and white hardcopies of the screen:

- "Black and White Line Styles" sets a white background color. All traces and information elements in the diagram areas are black, however, the traces are drawn in different line styles.
- "Black and White Solid" sets a white background color. All traces and information elements in the diagram areas are black. All traces are drawn with solid lines.

"User Define..." opens a dialog to modify the predefined schemes, changing the colors and styles of the individual display elements.

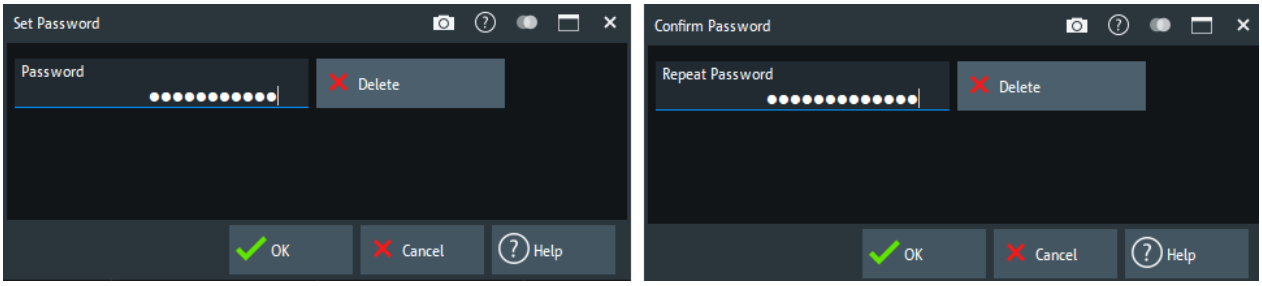
See Section 5.16.3.2, "Define User Color Scheme dialog", on page 803.

Remote command:
`SYSTem:DISPlay:COLor`

Hide Sensitive Information
Unmasks or masks all stimulus value occurrences in the **VNA GUI** for the current recall set.

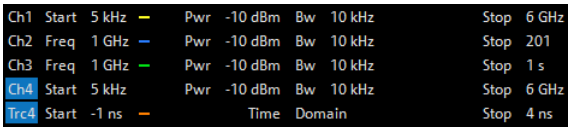
Note: This is a pure GUI feature. It does not protect you from any kind of data readout via remote control.

When you enable "Hide Sensitive Information" for a particular recall set (to mask the stimulus values), you are asked to set a password. If this password is non-empty, it is requested the next time someone tries to disable "Hide Sensitive Information" (to unmask all stimulus values) for this recall set.



Both state and password are stored in (and loaded from) the active recall set.

Channel Info
Shows or hides the channel lists in the lower part of the diagrams.



Remote command:
`DISPlay:ANNotation:CHANnel[:STATe]`

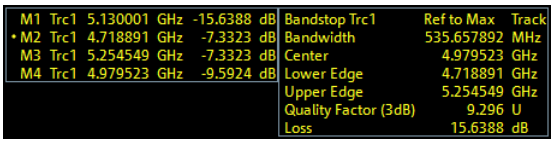
Trace Info
Shows or hides the trace lists in the upper part of the diagrams.



Remote command:
DISPlay:ANNotation:TRACe[:STATe]

Info Table: Show / Position
Shows or hides the info table and defines its position.

The info table is a possible container for info fields and can be placed to the bottom, to the left, or to the right of the screen. See also Section 5.7.8, "Info Field tab", on page 524.



Remote command:
DISPlay:ANNotation:INFO:STATe
DISPlay:ANNotation:INFO:POSition

Info Window
Shows/hides the Info Window

Remote command:
DISPlay:IWINdow[:STATe]

Font Size
Scales the fonts in the diagrams. The scaling affects the trace and channel lists, and the info fields.

Remote command:
DISPlay:RFSize

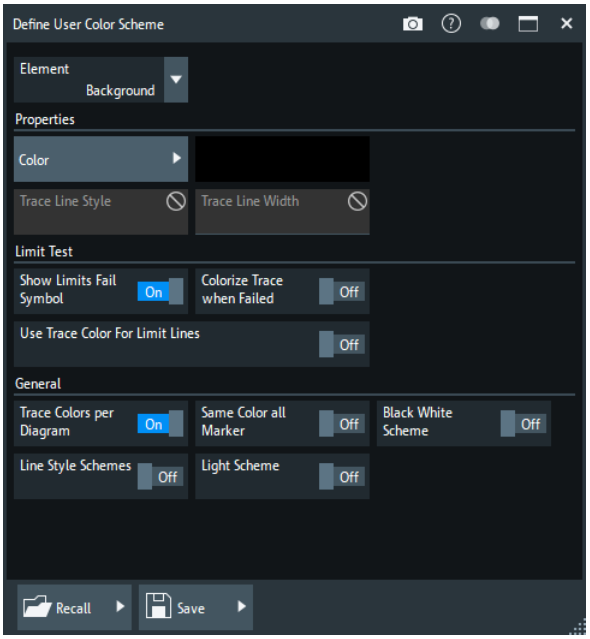
5.16.3.2 Define User Color Scheme dialog

The "Define User Color Scheme" dialog allows you to configure user-defined diagram color schemes for display and print. User-defined color schemes can be saved to a file for later reuse.

Access:

- System – [Display] > "Config" > "Define User Color..."
- Printer Setup dialog > "Print" tab > "Define Print Color..."

Related settings
Refer to "Color Scheme" on page 801.



Element
Selects the diagram element to be modified. The list contains the background and all traces (more precisely trace properties), text elements and lines in the diagrams.

The maximum number of trace properties can be configured in the "User Interface" tab of the "System Config" dialog (see "Number of Trace Colors" on page 816).

Remote command:
The `<DispEl>` suffix in the `DISPlay:CMAP<DispEl>...` commands identifies the screen element. See `DISPlay:CMAP<DispEl>:RGB`.

Properties
Configures the selected screen element.

- "Color" opens a standard color dialog where you can assign a color to the selected element.
- "Trace Line Style" and "Trace Line Width" are enabled if the selected element is a trace.

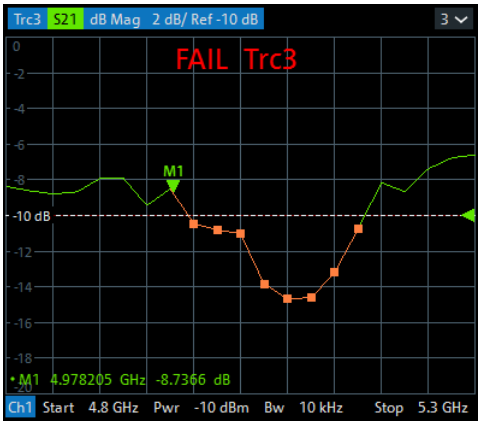
Remote command:
Display color scheme only
`DISPlay:CMAP<DispEl>:RGB`
`DISPlay:CMAP:TRACe:RGB`

Limit Test > Show Limit Fail Symbols
Displays or hides the colored squares on the trace indicating failed measurement points. Hide the squares if they cover too much of the trace. Instead of using the limit fail symbols, you can colorize the trace to highlight failed trace sections.

Remote command:
Display color scheme only
`DISPlay:CMAP:LIMit:FSYMbol[:STATe]`

Limit Test > Colorize Trace when Failed
Assigns a different trace color to failed trace segments.

The different color reaches from the last passed measurement point before the start of the failed segment to the last failed measurement point in the segment. Consequently, the colorized trace segment can begin before the begin of the failed range and can end before its end.



Remote command:
Display color scheme only
`DISPlay:CMAP:LIMit:FCOLorize[:STATe]`

Limit Test > Use Trc Color for Limit Lines
Assigns the trace color to all limit line segments associated with the trace. All other limit line color definitions are ignored.

Remote command:
Display color scheme only
`DISPlay:CMAP:LIMit[:STATe]`

General > Trace Colors per Diagram
Controls the color of traces that are created in a diagram or moved to another diagram.

- If "Trace Colors per Diagram" is enabled (factory preset), all diagrams use the same trace colors. The first trace in each diagram gets trace color 1, the second trace color 2 etc. (cyclically).
  In particular, if a trace is removed from a diagram, then all traces in this diagram that were created after the removed one, will change their color. And if this trace is moved to another diagram, its color typically changes as well.
- If "Trace Colors per Diagram" is disabled, then the traces are colored with trace colors 1 to 8 (cyclically) in the order they are created. No matter which diagram they are assigned to.
  All traces keep their original trace color (number).

Remote command:
Display color scheme only
`DISPlay:CMAP:TRACe:COLor[:STATe]`

General > Same Color all Markers
Selects a common marker color, which is independent of the trace colors.

Remote command:
Display color scheme only
DISPlay:CMAP:MARKer[:STATe]

General > Black White Scheme / Line Styles Scheme / Light Scheme
Modifies the user color scheme, in particular the trace and channel lines, in a predefined way. As an alternative, select predefined color schemes; see "Color Scheme" on page 801.

Remote command:
DISPlay:CMAP<DispEl>:RGB
DISPlay:CMAP:BWSCheme[:STATe]
DISPlay:CMAP:LSSCheme[:STATe]
DISPlay:CMAP:LSCHeme[:STATe]

Recall... / Save...
Opens standard dialogs to recall a previously saved color scheme or save the current scheme to a file. Color scheme files are non-editable files with the extension *.ColorScheme; the default directory is C:\Users\Public\Documents\Rohde-Schwarz\Vna\ColorSchemes.

Remote command:
Display color scheme:
MMEMory:STORe:CMAP
MMEMory:LOAD:CMAP
Print color scheme:
MMEMory:STORe:CMAP:HCOPy
MMEMory:LOAD:CMAP:HCOPy

5.16.3.3 Info Window

The "Info Window" displays a configurable set of marker properties and Bandfilter search results. It automatically resizes the displayed text to fit the window height and width.

Access: System – [Display] > "Config" > "Info Window" control



Use the "Configuration..." button to open the Info Window Configuration dialog and select the information items to be displayed.

To disambiguate the displayed marker info fields, you can assign descriptive names to the markers (see "Marker Name" on page 506).

If a trace is hidden, then its marker and bandfilter search info is also hidden in the "Info Window".

Info Window Configuration dialog

The "Info Window Configuration" dialog allows you to select the markers and Display elements of a diagram results to be displayed in the Info Window.

Access: Info Window > "Configuration..."



Use the "Info Field" combo box to select a marker or bandfilter search info field whose contents you want to display in parts or in total. It contains all info fields of the current recall set that are currently not displayed in the info table (see "Info Table: Show / Position" on page 803).

Content Selection

The lower part of the "Info Window Configuration" dialog allows you to pick the contents of the selected info field you want to display in the Info Window.

For marker info fields, use the "Add" and "Remove" buttons to show/hide the selected marker in the "Info Window".



Figure 5-95: Content selection: marker info fields

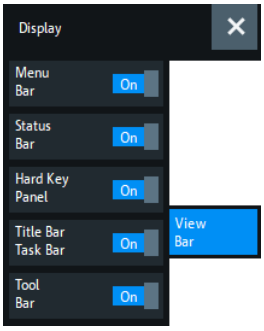
For Bandfilter search info fields, simply pick the results to be displayed.

Remote command:

```
DISPlay:IWINdow:MARKer<Mk>[:STATe]
DISPlay:IWINdow:BFILter[:STATe]
```

5.16.4 View Bar tab

Displays or hides information panels and bars of the graphical user interface. Hiding the information elements leaves more space for the diagrams. All elements can be shown or hidden simultaneously.



Menu Bar
Toggles the visibility of the "Menu Bar".

For background information, see Section 3.3.2.5, "Menu bar", on page 52.

Remote command:
`SYSTem:DISPlay:BAR:MENU[:STATe]`

Status Bar
Toggles the visibility of the "Status Bar".

For background information, see Section 3.3.2.6, "Status bar", on page 53.

Remote command:
`SYSTem:DISPlay:BAR:STATus[:STATe]`

Hard Key Panel
Toggles the visibility of the "Hard Key Panel".

For background information, see Section 3.3.2.3, "Hardkey panel", on page 50.

The "Hard Key Panel" can also be closed via the "X" button in its top-right corner.

Remote command:
`SYSTem:DISPlay:BAR:HKEY[:STATe]`

Title Bar Task Bar
Toggles the visibility of the title bar of the VNA application window and the Windows® task bar.

If disabled (default) the VNA application is displayed in full screen mode with invisible title bar and Windows® task bar. If enabled, it is displayed as a regular window.

For background information, see Section 3.3.2.1, "Title bar", on page 49.

Remote command:
`SYSTem:DISPlay:BAR:TITLe[:STATe]`

Tool Bar
Toggles the visibility of the "Tool Bar".

For background information, see Section 3.3.2.2, "Toolbar", on page 49.

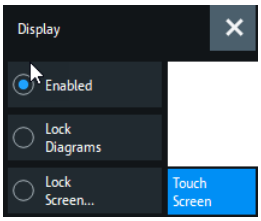
Remote command:
`SYSTem:DISPlay:BAR:TOOLs[:STATe]`

Additional Function: Minimize/Mazimize the Softtool Panel
The softtool panel can be minimized/maximized via the "X"/hamburger icon in its top-right corner.

Remote command:
`SYSTem:DISPlay:BAR:STOols[:STATe]`

5.16.5 Touchscreen tab

Allows you to lock the touchscreen functionality of a R&S ZNB to prevent inadvertent entries.



Enabled / Lock Diagrams / Lock Screen
- "Enabled" – touchscreen control of the R&S ZNB fully enabled. All control elements are active.
- Lock diagrams – drag and drop functions in the diagrams are disabled, all other control elements (e.g. the softtool panels) are still active.
- Lock screen – all control elements are locked. Pressing any front panel key on the analyzer (or sending `SYSTem:TSLock OFF`) re-enables touchscreen control.

Remote command:
`SYSTem:TSLock`

5.17 Setup softtool

The "Setup" softtool allows you to define various system-related settings, to manage global resources, to get system information and to execute service functions.

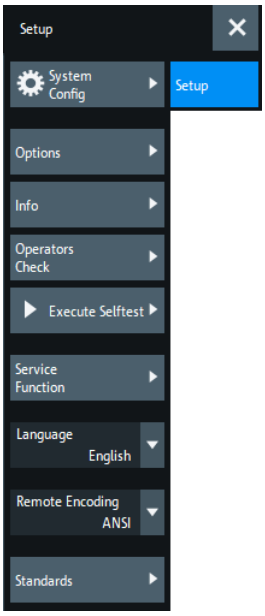
Access: System – [Setup]

Persistent vs. session settings
The settings in the "Setup" softtool and the configuration dialogs are global settings and not affected by a "Preset" or shutdown of the analyzer.

5.17.1 Setup tab

Gives access to system-wide properties, settings, resources and service functions.

5.17.1.1 Controls on the Setup tab



The following buttons in the "Setup" tab open related dialogs:

- **"System Config":** System Config dialog
- **"Options":** "Info" dialog > Options tab
- **"Info":** "Info" dialog > Setup tab
- **"Operators Check...":** Operators Check wizard
- **"Execute Selftest...":** "Info" dialog > Selftest tab
  (Runs the self-test first)
- **"Service Function":** Service Function dialog

Language
Selects the language of the graphical user interface. A message box indicates that the vector network analyzer application needs to be restarted to activate a different language.

Remote Encoding
Selects the character encoding used at the remote interface. The selected encoding applies to directory and filenames, calibration kit names, calibration unit characterizations and display titles.

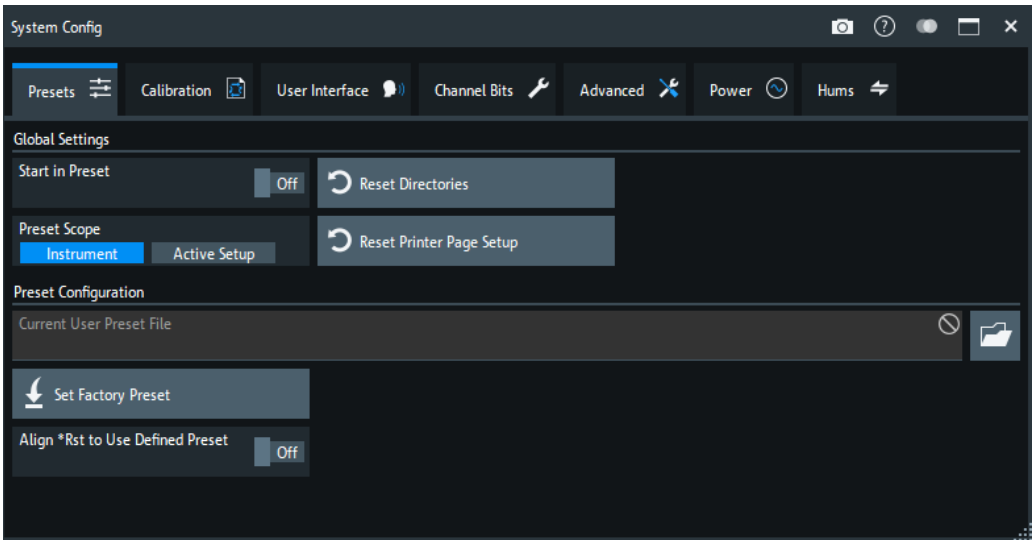
Currently the following encodings are supported: ANSI (default), UTF-8, Shift JIS.

Remote command:
`SYSTem:COMMunicate:CODec`

5.17.1.2 System Config dialog

The "System Config" dialog allows you to define global settings that are not affected by an instrument reset. See Section 4.1.1, "Global (persistent) settings", on page 79.

Access: System – [Setup] > "Setup" > "System Config"



- Presets tab 811
- Calibration tab 813
- User Interface tab 815
- Channel Bits tab 817
- Advanced tab 818
- Power tab 820
- HUMS tab 823

Presets tab

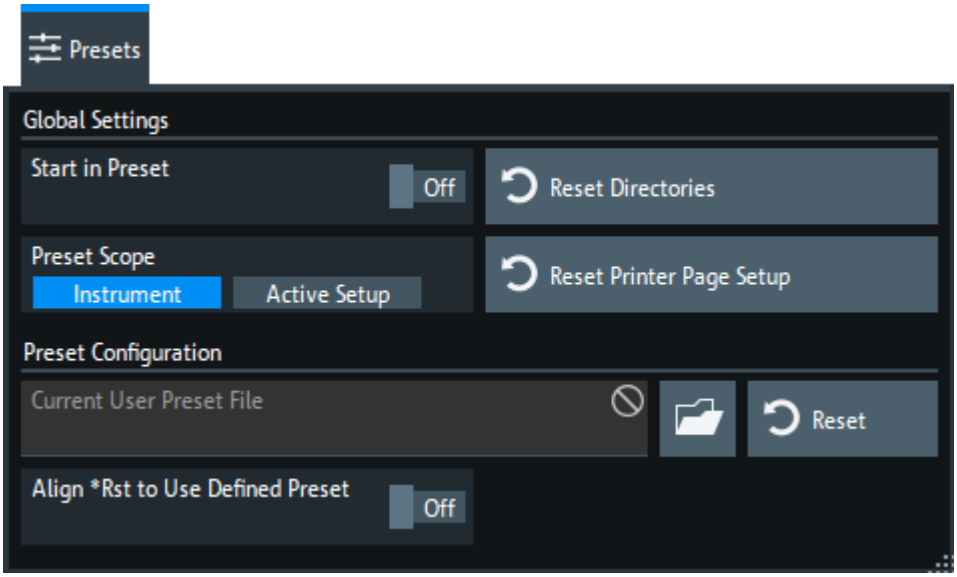
Specifies the behavior of the R&S ZNB upon a preset.

Preset vs. factory reset

A "Preset" replaces the current measurement setups by a setup loaded from a predefined or user-defined recall set. Global (persistent) settings are not affected.

To reset the global settings, you can use the "Reset Factory State" script that can be accessed from the instrument's program group in the Windows "Start" menu.

This script can be helpful in case the firmware does not start. It also resets the configured "Preset" functionality to the default (predefined) recall set.



Start in Preset
If this option is active, the R&S ZNB FW always starts with the configured Preset Configuration. Otherwise it starts with the previous state, i.e. the settings that were persisted during the previous shutdown.

Remote command:
`SYSTem:PRESet:STARt`

Preset Scope
Defines whether a preset affects all open recall sets ("Instrument") or the active recall set only.

The "Preset Scope" applies to the GUI and to the `SYSTem:PRESet[:DUMMy]` command. The `*RST` command always resets all open recall sets.

Remote command:
`SYSTem:PRESet:SCOPe`

Reset Directories/Reset Printer Page Setup
The two buttons reset all directory settings (e.g. the directories for storing trace data, limit lines, calibration data...) and all settings in the "Printer Setup" dialog to default values. See Section 5.14.4, "Printer Setup dialog", on page 785.

Remote command:
N.a.

Preset Configuration
Specifies whether the preset action performs a factory preset or restores a user preset file. A user preset file is an arbitrary recall set file (`*.znx|*.znxml`), that can be created from the active setup using System – [File] > "Save...". If the current user preset file is not found (e.g. because it was deleted or moved), System – [Preset] initiates a factory preset.

The preset configuration applies to:
- The System – [Preset] hardkey

- The remote commands `*RST` and `SYSTem:PRESet` (if "Align *RST to Use Defined Preset" is selected in the Align *Rst to Use Defined Preset section).

Furthermore, the user preset file is used as a template, whenever New recall set files are created from the GUI.

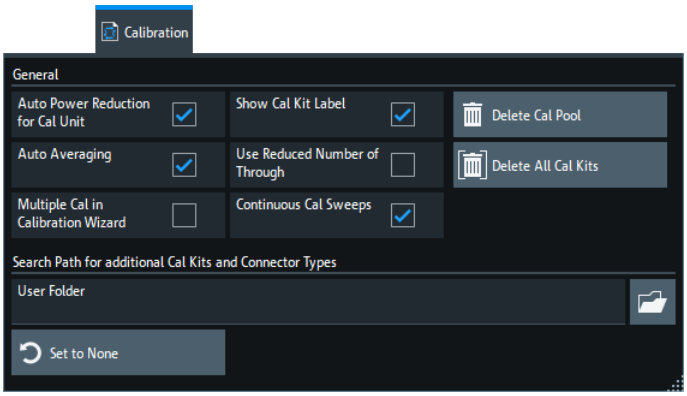
Remote command:
`SYSTem:PRESet:USER:NAME`
`SYSTem:PRESet:USER[:STATe]`

Align *Rst to Use Defined Preset
Determines the behavior of the `*RST` and `SYSTem:PRESet` commands. If enabled, and if a valid setup file is selected in the Preset Configuration section, then they perform a user-defined preset. Otherwise, they perform a factory preset.

Remote command:
`SYSTem:PRESet:REMote[:STATe]`

Calibration tab

Provides general system error correction (calibration) settings.



Auto Power Reduction for Cal Unit
Limits the power in the calibrated reference plane to -10 dBm while an automatic calibration is performed. Applying this source power to the ports of the calibration unit ensures best accuracy of the automatic calibration. The source powers are reset to their original values after the calibration is completed. The automatic power reduction can be deactivated in case that the test setup introduces a large attenuation.

Remote command:
`SYSTem:COMMunicate:RDEVice:AKAL:PREDuction[:STATe]`

Auto Averaging
Activates automatic averaging, which means that the VNA performs multiple calibration sweeps and applies averaging to reduce trace noise. In contrast to regular averaging (see Section 5.9.4, "Average tab", on page 534), the number of calibration sweeps is calculated automatically.

Remote command:
`[SENSe:]CORRection:COLLect:AVERage`

Multiple Cal in Calibration Wizard
Enables/disables multiple calibrations in the calibration wizard, which allows to set up and perform multiple calibrations in one manual calibration procedure. See Section 5.11.1.4, "Calibration Unit wizard (MultiCal)", on page 576, Section 5.11.1.6, "Calibration Setting wizard (MultiCal)", on page 591, Section 5.11.1.9, "SMARTerCal wizard (Cal Unit, MultiCal)", on page 612 and Section 5.11.1.11, "SMARTerCal wizard (manual, MultiCal)", on page 627).

Remote command:
`SYSTem:DISPlay:DIALogs:SETup:MCAL[:STATe]`

Show Cal Kit Label
Enables/disables the "Calibration Info" dialog during manual calibration (see "Start Cal Sweep" on page 589).

Independent of the state of the "Show Cal Kit Label" flag, cal kit labels are displayed in several other manual calibration dialogs.

Use Reduced Number of Throughs for Cal Unit
Enables or disables the Reduced Through logic for automatic full n-Port calibrations.

If unchecked (= factory preset), it is possible to measure **all** Through connections during an automatic calibration.

Remote command:
`[SENSe:]CORRection:COLLect:AUTO:MCONnect`

Delete Cal Pool / Delete All Cal Kits
Deletes all calibration data and all cal kit data. See Section 5.11.4.3, "Calibration Manager dialog", on page 657.

Remote command:
n/a

Search Path for additional Cal Kits and Connector Types
Contains the name and path of a special directory for cal kit files (`*.calkit`). All cal kit files in the special directory are loaded automatically as predefined kits (i.e. read-only kits that cannot be modified) every time the VNA application is started. It is possible to select the default cal kit directory
`C:\Users\Public\Documents\Rohde-Schwarz\Vna\Calibration\Kits` or any other directory. "None" means that no additional cal kit files are loaded on startup.

Use the special directory to make sure that you do not have to import kits manually, even after terminating the VNA application improperly. In this case, previously imported cal kit files are not stored in the recall set file.

Remote command:
`MMEMory:LOAD:CKIT:UDIRectory`

Continuous Cal Sweeps
Disables/enables continuous calibration sweeps.

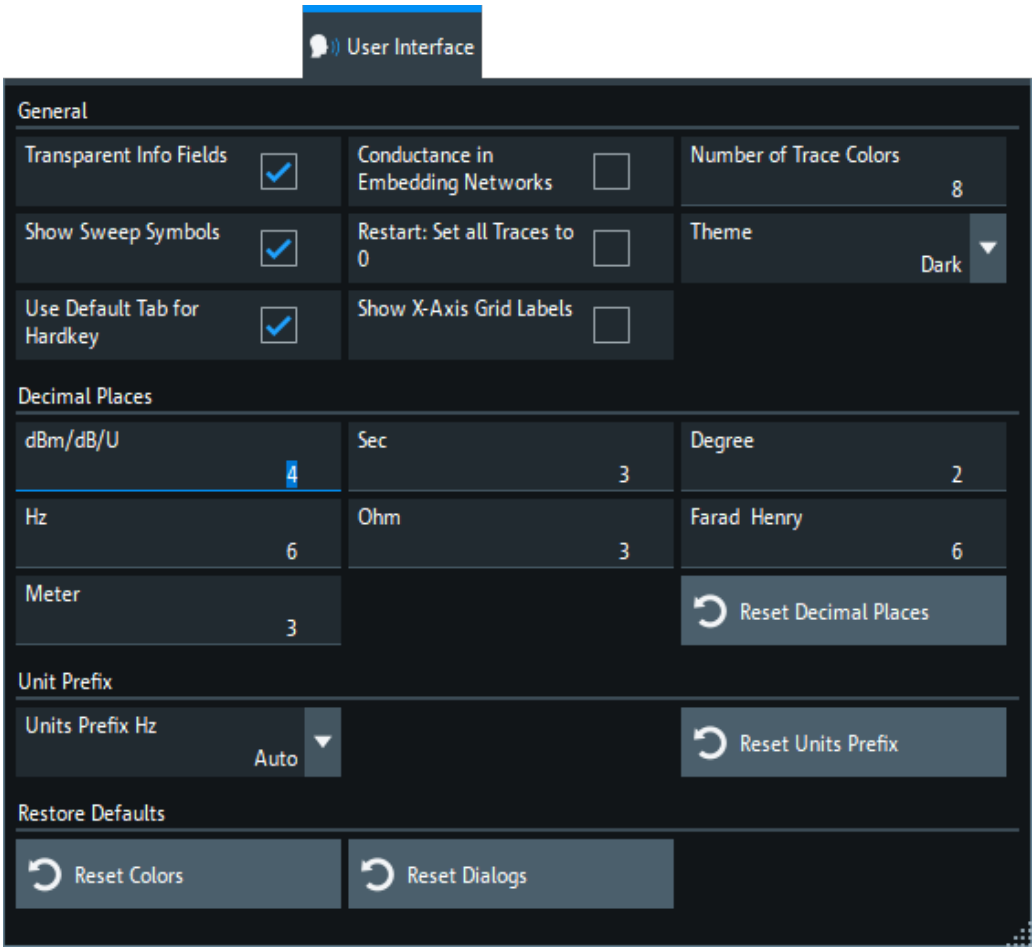
Can be configured here and from the calibration wizards.

Disabling continuous calibration sweeps can be beneficial when calibrating mutiple channels, and switching between these channels involves mechanical switches.

Remote command:
[SENSe:]CORRection:COLLect:SWEep:CONTinuous

User Interface tab

Provides general user interface configurations.



Transparent Info Fields
Enables or disables transparent info fields for markers and trace statistics.

Transparent info fields do not hide an underlying trace.

Remote command:
n.a.

Show Sweep Symbols
Turns sweep symbols on or off.

Sweep symbols are arrows pointing downward onto the trace. They are displayed if the sweep time exceeds an upper limit, e.g. if the number of points is high or the measurement bandwidth is low.

Remote command:
n.a.

Use Default Tab for Hardkey
If enabled (system default), the Function keys activate the first enabled tab of their associated softtool. Otherwise the last used tab is activated.

For background information, see Section 5.1, "Function keys and softtools", on page 289.

Remote command:
N.a.

Conductance in Embedding Networks
Changes the presentation of "capacitance C<i> in parallel with resistance R<i>" circuit blocks in lumped de/embedding networks (see Section 4.6.2.3, "Circuit models for 2-port networks", on page 205 and Section 4.6.2.4, "Circuit models for 4-port networks", on page 207).

If active, the resistance R<i> is displayed and specified as conductance G<i> (=1/R<i>).

Remote command:
`SYSTem:DISPlay:CONDuctances`n.a.

Restart: Set all Traces to 0
Defines the behavior of the "Restart Sweep" function (see "Restart Sweep" on page 560).

If disabled (factory default) the previous measurement data are preserved. Otherwise, they are deleted.

Remote command:
`SYSTem:TRESet[:STATe]`

Show X-Axis Grid Labels
If enabled, the firmware displays X-axis grid labels in Cartesian diagrams with linear scale.



Remote command:
`SYSTem:DISPlay:XLABels`

Number of Trace Colors
Defines the maximum number of trace colors (trace properties for the user defined color scheme).

Remote command:
`SYSTem:DISPlay:TRACes:CCOunt`

Decimal Places
Defines the number of fractional digits for quantities with different physical units. The settings affect entries and results, e.g. the values in the marker lists.

Note: If your instrument is equipped with option R&S ZNB-K19, 1 mHz Frequency Resolution, set "Decimal Places" of unit "Hz" to *12* to utilize the high frequency resolution.

Remote command:
N.a.

Units Prefix
Sets the unit prefix for frequencies (Base unit: Hz) to kilo (k), mega (M), giga (G) or tera (T) or lets the R&S ZNB select the appropriate prefix ("Auto" = default setting).

Remote command:
N.a.

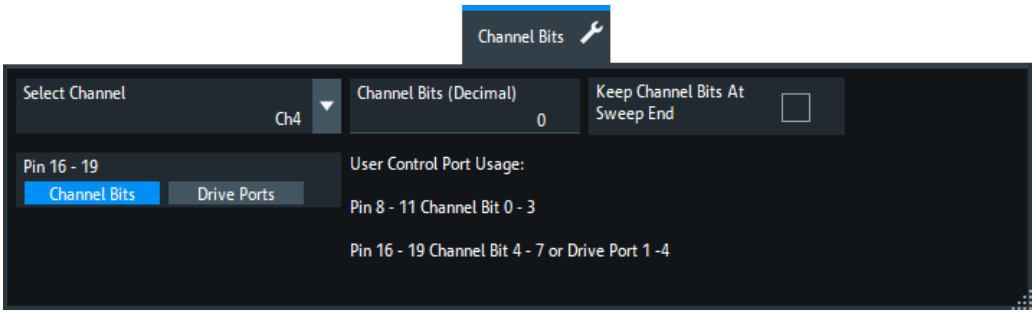
Reset Colors / Reset Dialogs / Reset Decimal Places / Reset Units Prefix
Resets the color settings ("System" > "Display" < "Config"), the dialog properties, the "Decimal Places" and the "Units Prefix" settings. These settings are global and not affected by an instrument preset.

Remote command:
N.a.

Channel Bits tab

Sets a channel-dependent 8-bit decimal value (0, ..., 255) to control eight independent output signals at the USER PORT connector (lines 8 to 11 and 16 to 19).

Setting the channel bits does not change the analyzer state.



Channel Bits (Decimal)
Entry of the 8-bit decimal value (0 ... 255) for the selected channel. The channel bits control eight output signals at the USER PORT connector. The signals are 3.3 V TTL signals which can be used to differentiate between up to 256 independent analyzer states. For an application example, refer to the description of the remote-control command.

The decimal values have the following effect:

- 0 means that no output signals are enabled at any of the pins 8, 9, 10, 11 and 16, 17, 18, 19.
- 1 enables the output signal at pin 8. The signal is switched on while a measurement sweep is running in the selected channel. All other signals are inactive.
- 2 enables the output signal at pin 9.
- 3 enables the output signals at pins 8 and 9.
- ...
- 255 enables the output signals at all pins. See also "Pin 16 - 19" on page 818.

Remote command:
`OUTPut<Ch>:UPORt[:VALue]`

Keep Channel Bits At Sweep End
By default, at the end of a channel sweep the channel bits are reset. Activating this setting allows you to keep their state.

Note that this setting is valid for **all** channels.

Remote command:
`OUTPut:UPORt:KEEP`

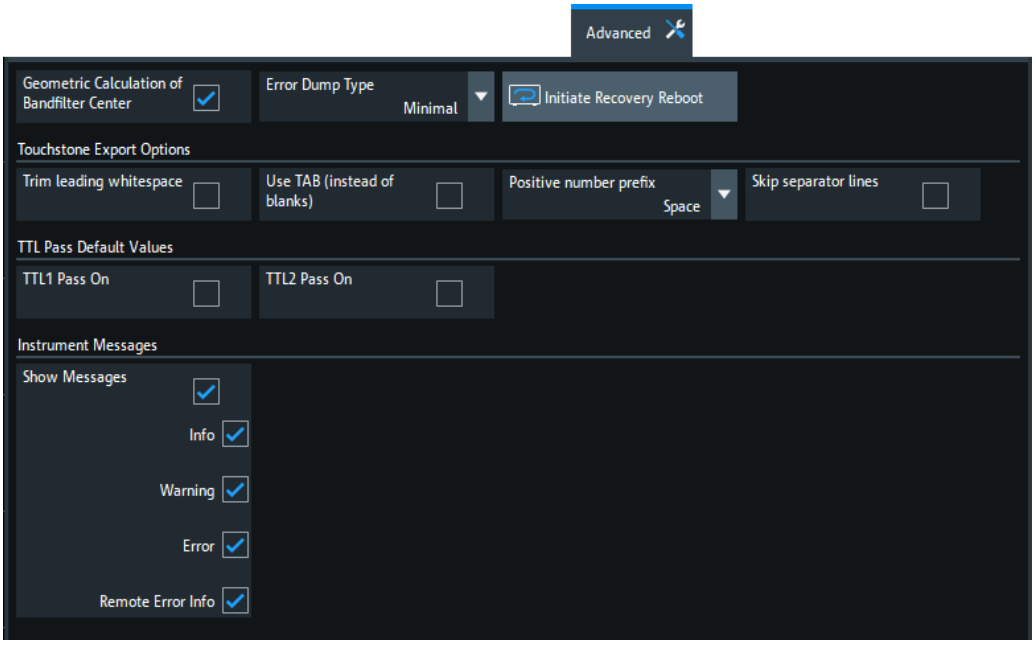
Pin 16 - 19
Selects the control mechanism for the signals at pins 16 to 19 of the USER PORT connector.

- **"Channel Bits":** Signals are controlled by channel bits 4 to 7. No drive port indication at the USER PORT connector.
- **"Drive Ports":** Signals indicate the active drive ports. The number of active channel bits is reduced to 4 (pins 8 to 11).

Remote command:
`OUTPut:UPORt:ECBits`

Advanced tab

Collects several advanced settings.



Geometric Calculation of Bandfilter Center
Defines how bandfilter searches calculate the center frequency of the passband or stopband (see "Bandfilter search" on page 102).

If "Geometric Calculation of Bandfilter Center" is enabled, the *geometric mean* of the lower band edge and upper band edge frequencies is used, otherwise their *arithmetic mean*.

Remote command:
`CALCulate:MARKer:FUNCtion:BWIDth:GMCenter`

Error Dump Type
Determines the level of detail ("Minimal", "Normal", "Large", "Full") and hence the size of the dump files that are created if a firmware exception occurs. "None" disables dump file creation.

The latest 5 exception dumps can be retrieved using the "Save Report" function of the "Info" dialog (see "Create R&S Support Information" on page 828).

For further information, see Section 9, "Error messages and troubleshooting", on page 1687.

Remote command:
`DIAGnostic:DUMP:SIZE`

Initiate Recovery Reboot
Allows you to reboot the instrument and launch the "R&S Recovery Environment". From there, you can restore the original system image (including operating system and firmware).

See Section 13.2, "System recovery", on page 1701.

Touchstone Export Options
Configures whitespace insertion during legacy Version 1.1 (ZNx) Touchstone file export.

The default export format is explained in Section 4.4.2.1, "Touchstone files", on page 144:

- Logical columns are vertically aligned using spaces
- Positive and negative numbers are vertically aligned by prefixing positive numbers with blanks
- The frequencies are horizontally separated from the corresponding S matrices using leading spaces
- The content parts (header, S matrices for different frequencies) are separated by blank lines

Use TAB (instead of blanks) ← Touchstone Export Options
If enabled, columns are separated by tabs rather than spaces.

Remote command:
`MMEMory:STORe:TRACe:OPTion:TABS`

Trim Leading Whitespace ← Touchstone Export Options
If enabled, whitespace at the beginning of each line is removed.

Remote command:
`MMEMory:STORe:TRACe:OPTion:TRIM`

Positive Number Prefix ← Touchstone Export Options
Positive numbers can either be prefixed by blanks, by plus signs or not at all.

Remote command:
`MMEMory:STORe:TRACe:OPTion:PLUS`

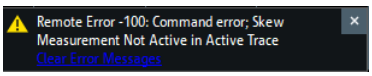
Skip Separator Lines ← Touchstone Export Options
If enabled, the content parts are no longer separated by blank lines.

Remote command:
MMEMory:STORe:TRACe:OPTion:SSEParator

TTL Pass Default Values
Defines the default values of TTL1 Pass / TTL2 Pass for new traces.

Remote command:
SYSTem:TTLout<Pt>:STATus[:STATe]

Instrument Messages
Defines which instrument events are indicated by an information popup.



Display of information popups can be globally disabled or limited to certain event types.

Show Messages ← Instrument Messages
Defines whether information popups are displayed at all.

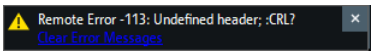
Remote command:
SYSTem:ERRor:DISPlay:STATe

Info / Warning / Error ← Show Messages ← Instrument Messages
Selectively disables/enables display of information popups for the related event type.

If information popups are globally switched off (Instrument Messages disabled), these buttons are disabled. However, their enabled state is memorized.

Remote command:
SYSTem:ERRor:DISPlay:INFO
SYSTem:ERRor:DISPlay:WARNings
SYSTem:ERRor:DISPlay:ERRor

Remote Error Info ← Show Messages ← Instrument Messages
Defines whether information popups are displayed whenever a remote control command error occurs.



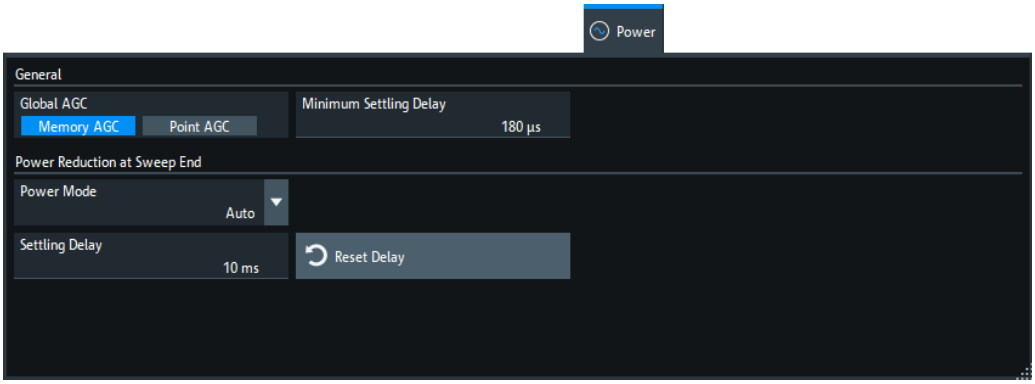
The displayed information can be useful for program development and optimization; it does not necessarily indicate that a remote control script is faulty or non-executable.

Note
- If either "Show Instrument Messages" or "Show Info Messages" is disabled, this button is disabled. However, its enabled state is memorized.
- For SCPI error `-113, Undefined header`, no tooltip is displayed.

Remote command:
SYSTem:ERRor:DISPlay[:REMote]

Power tab

The settings in this tab define the global power settings of the VNA.



Global AGC
Allows you to select between two AGC algorithms:

"Memory AGC" (factory default)	Optimized gain control pre-measurement: only if necessary and for as few sweep points as possible.
"Point AGC"	Legacy AGC algorithm: gain control pre-measurement at every sweep point. Typically significantly slower than "Memory AGC".

Remote command:
`SYSTem:AGC:GLOBal:METHod`

Minimum Settling Delay
Defines the minimum settling delay **for external generators**. The delay is enforced:
- Whenever settings are applied to a driving generator port, i.e. as a minimum delay between the setting and the actual measurement.
- Irrespective of the sweep type, and whether the generator actually sweeps

Factory default is 180 µs.

Remote command:
`SOURce:POWer:GENerator:SDELay`

Power Reduction at Sweep End
The power reduction settings apply to all sweep modes but are particularly useful in single sweep mode.

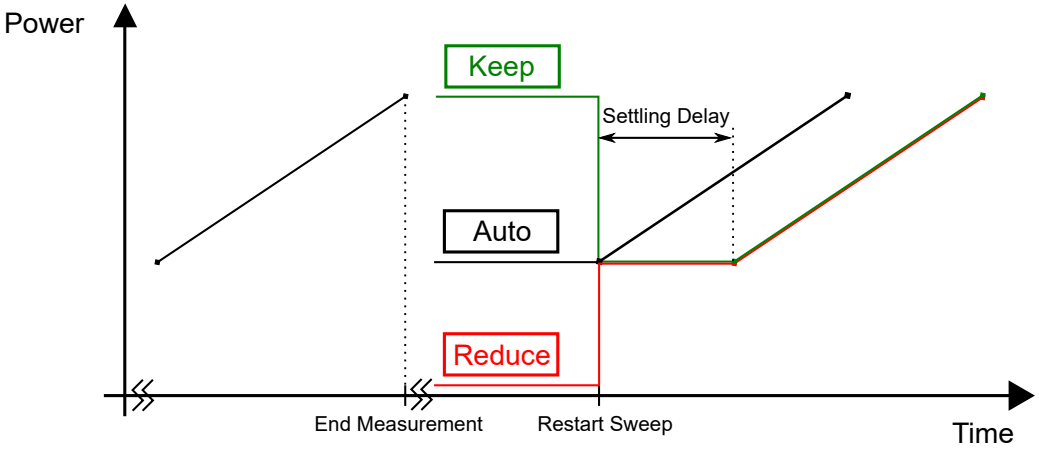


Figure 5-96: Power control at sweep end in Single Sweep mode (power sweep)

Note: By default, at sweep end, the output power of the first measurement point is restored ("Auto" mode). After a small settling time (~10 ms), the sweep can be restarted.
Selecting a different behavior can result in a significantly longer total measurement time, in particular if a long "Settling Delay" is used. Do not forget to revert the Power Mode at Sweep End to "Auto" mode if it is not necessary to "Reduce" or "Keep" the power at the sweep end. A preset is not sufficient for this purpose.

Power Mode at Sweep End ← Power Reduction at Sweep End
The analyzer offers three power modes at sweep end:

- "Auto" results in the shortest measurement time (default setting). If enabled, at sweep end the output power of the first measurement point is restored. The configured Settling Delay is not applied.
- "Reduce" is intended for measurements on sensitive DUTs (primarily: power sweeps). If enabled, at sweep end the output power of the driving port is reduced as if the channel base power was set to its minimum possible value. The configured settling delay is applied.
- "Keep" is intended for power sweeps. If enabled, at sweep end the output power of the last measurement point is kept. The configured settling delay is applied.

See Figure 5-96.

Note:

- Compared to "Auto" mode, "Reduce" and "Keep" can result in significantly longer measurement times - in particular if an extended settling delay is used.
- The output power is not altered if there is only a single channel with a single driving port, performing a "Time" or "CW Mode" sweep.
- In triggered mode, the analyzer always uses the settings of the first measurement point while waiting for the trigger signal.

Remote command:
`SOURce:POWer:SWEepend:MODE`

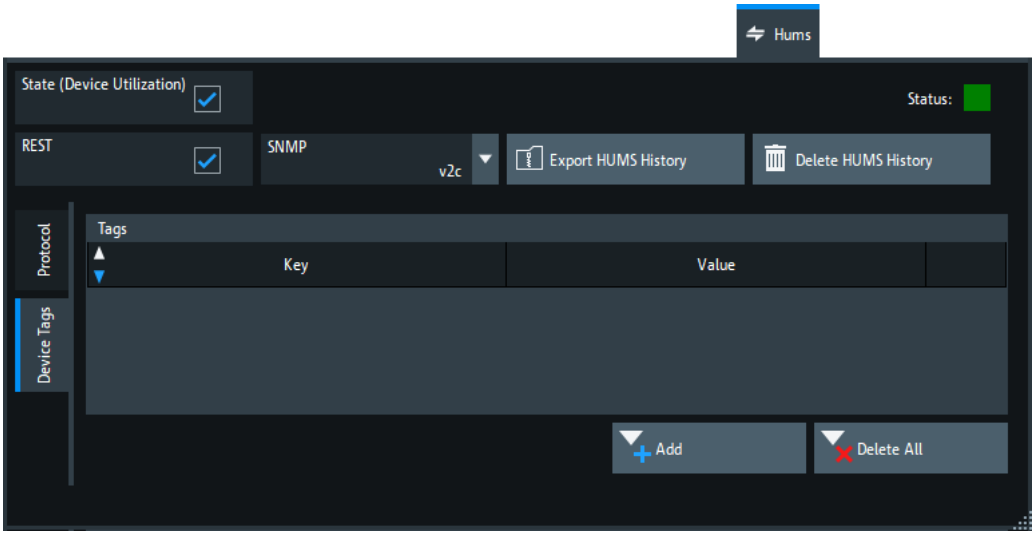
Settling Delay / Reset Delay ← Power Reduction at Sweep End
If Power Mode at Sweep End is set to "Reduce" or "Keep", the "Settling Delay" defines the time between Restart Sweep request and sweep start. See Figure 5-96 for an illustration.

Use the "Reset Delay" button to adjust the "Settling Delay" to its default value.

Remote command:
`SOURce:POWer:SWEepend:SDELay`

HUMS tab

The "HUMS" tab allows you to enable and configure R&S HUMS (see Section 4.7.13, "Health and utilization monitoring service (HUMS)", on page 257). It is only available if software option R&S ZNB-K980 is installed.



Common controls

State (Device Utilization)/Status
Enables/disables the HUMS service.

The "Status" LED indicates the current state of the HUMS service:
- Gray: inactive
- Green: up and running

If HUMS has been used before, turning on restores the previous protocol settings.

Remote command:
`DIAGnostic:HUMS:STATe`

REST ← State (Device Utilization)/Status
Enables/disables the REST interface to the HUMS service.

Remote command:
`SYSTem:COMMunicate:REST:ENABle`

SNMP ← State (Device Utilization)/Status
Enables/disables the SNMP interface to the HUMS service and allows you to select the supported SNMP protocol versions.

Depending on the enabled versions, different parts of the Protocol subtab are enabled.

Note: If you activate v2c, version v1 is also active.

Remote command:
`SYSTem:COMMunicate:SNMP:VERSion`

Delete HUMS History
Allows you to delete the previously collected health and monitoring information from the instrument.

Remote command:
`DIAGnostic:HUMS:DELete:ALL`

Export HUMS History
Allows you to export the previously collected health and monitoring information to a zipped JSON file.

Remote command:
`DIAGnostic:HUMS:SAVE`

Protocol subtab

Gives access to SNMP configuration options for protocol version 2c and 3.

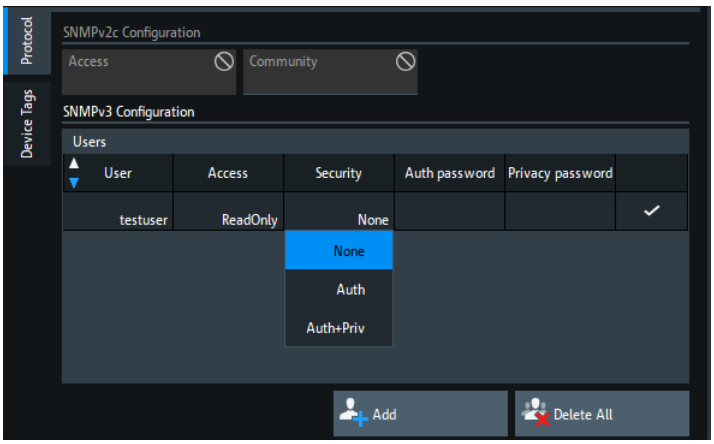
SNMPv2c Configuration
Allows you to grant read or read-write access via SNMPv2c, and to specify the related community string.



An SNMP community represents a collection of devices and agents grouped to monitor them. Authorized managers and the managed devices belong to an SNMP community. You can specify different communities for read-write and read-only access.

Remote command:
`SYSTem:COMMunicate:SNMP:COMMunity:RO`
`SYSTem:COMMunicate:SNMP:COMMunity:RW`

SNMPv3 Configuration
Allows you to enable and configure the SNMPv3-specific authentication and privacy features (passwords for access security and data encryption).



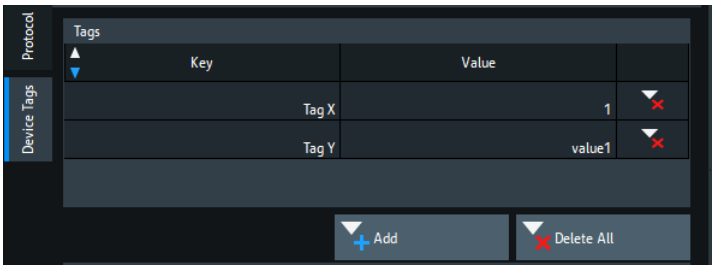
- To add a new user, select the "Add" button, enter the data and create the user by selecting the "✓" icon.
- To delete a single user, select the trash icon in the corresponding row.
- To delete all users, select the "Delete All" button.

Remote command:

```
SYSTem:COMMunicate:SNMP:USM:USER:ALL?
SYSTem:COMMunicate:SNMP:USM:USER
SYSTem:COMMunicate:SNMP:USM:USER:DELete
SYSTem:COMMunicate:SNMP:USM:USER:DELete:ALL
```

Device Tags subtab

Allows you to manage key-value pairs for HUMS device tagging.



A device tag is a label to assign to your instrument. You can create any device tag for your instrument and define it by a specific key and value.

Add / Delete (Trash) / Delete All

- To add a new device tag, select the "Add" button and enter the "Key" and "Value strings".
  The "Key" must be unique.
- To delete a single device tag, select the trash icon in the corresponding row.
- To delete all device tags, select the "Delete All" button.

Remote command:

```
DIAGnostic:HUMS:TAGS:ALL?
DIAGnostic:HUMS:TAGS[:VALue]
```

```
DIAGnostic:HUMS:TAGS:DELete
DIAGnostic:HUMS:TAGS:DELete:ALL
```

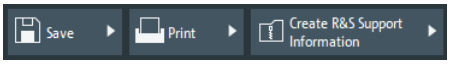
5.17.1.3 Info dialog

The "Info" dialog displays information about the instrument and its operation. All functions are primarily intended for error diagnostic and service purposes; see Section 9.3, "Collecting information for technical support", on page 1691. Many "Info" tabs also display softkeys for printing the contents or saving them to a file.

Access: System – [Setup] > "Setup" > "Info"

Common controls in the Info dialog

The "Save...", "Print...", and "Save Report" buttons at the bottom of the "Info" dialog allow you to save the contents of the open tab to a file or to create a hardcopy.

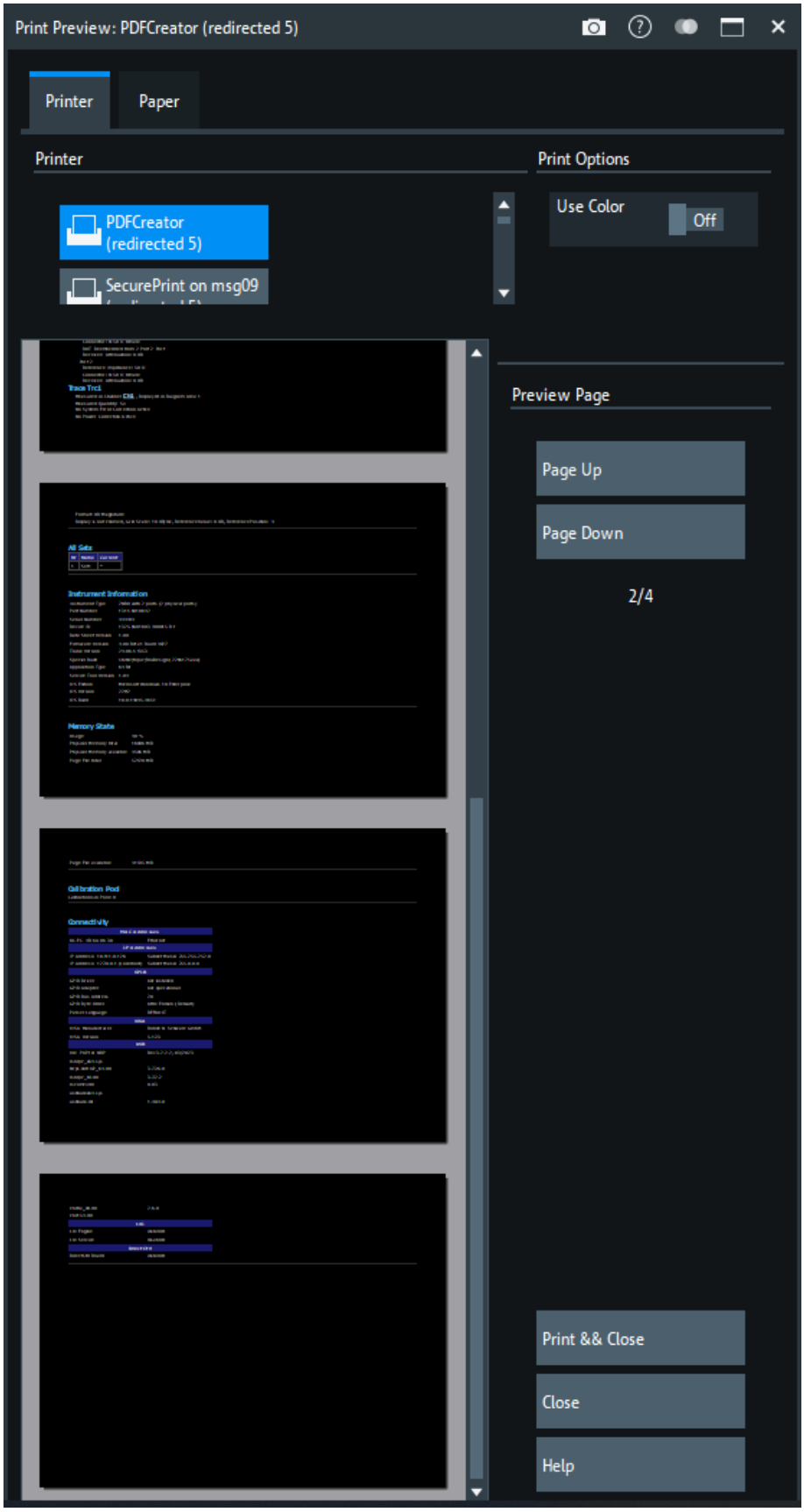


Save...
Allows you to save the contents of the open tab to an HTML file.

Remote command:

```
DIAGnostic:DEVice:STATe
SYSTem:DFPRint?
```

Print...
Allows you to print the contents of the open tab. Opens a "Print Preview" dialog for printer and paper setup.


Print Preview: PDFCreator (redirected 5)
Printer
Paper
Printer
PDFCreator (redirected 5)
SecurePrint on msg09
Print Options
Use Color
Off
Preview Page
Page Up
Page Down
2/4
Print && Close
Close
Help

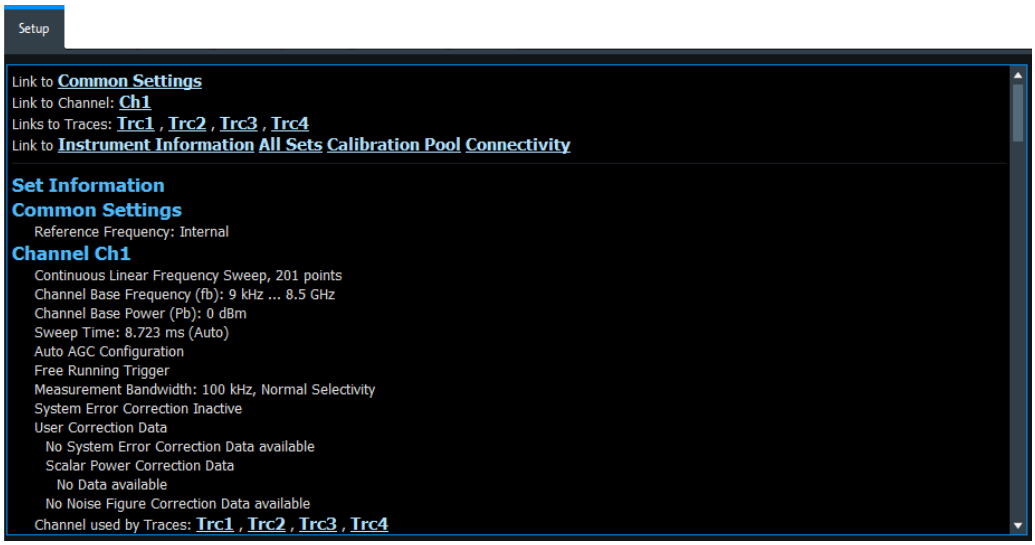
Create R&S Support Information
Saves the current self-test results to a zipped report file that you can send to R&S Support for fault diagnosis; see Section 9.3, "Collecting information for technical support", on page 1691.

Since FW V4.21, the "R&S Support Information" zip report also contains the current (unsaved) setup.

Remote command:
`DIAGnostic:DEVice:STATe`
`SYSTem:DFPRint?`

Setup tab

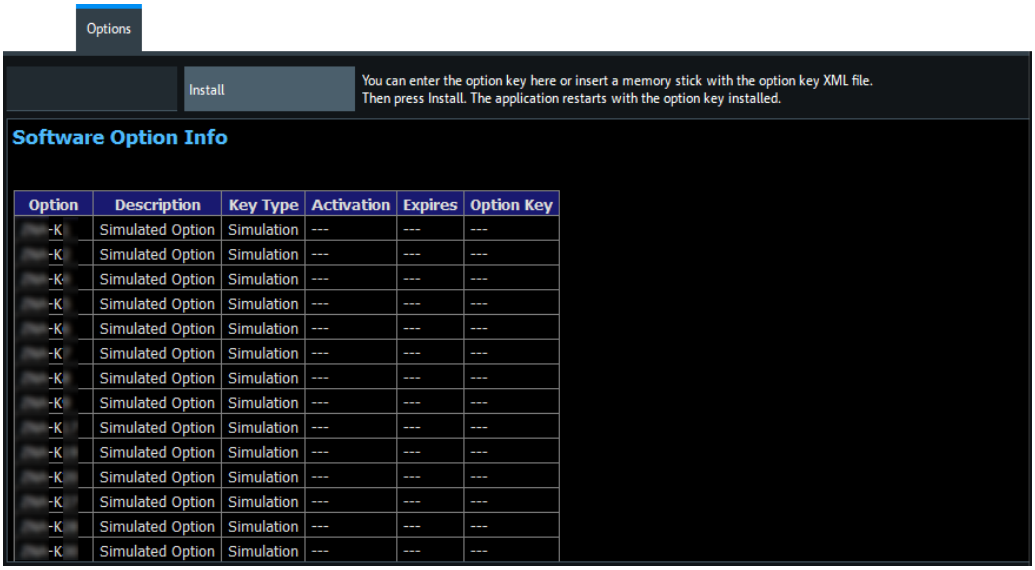
Displays the channel and trace settings of the active recall set and the main characteristics of the instrument, including its IP address.



Options tab

Shows the installed software and hardware options. You can also enable additional software options using the option key supplied with the option. Proceed according to the instructions in the dialog.

For an overview of options, refer to Section 4.7, "Optional extensions and accessories", on page 216.

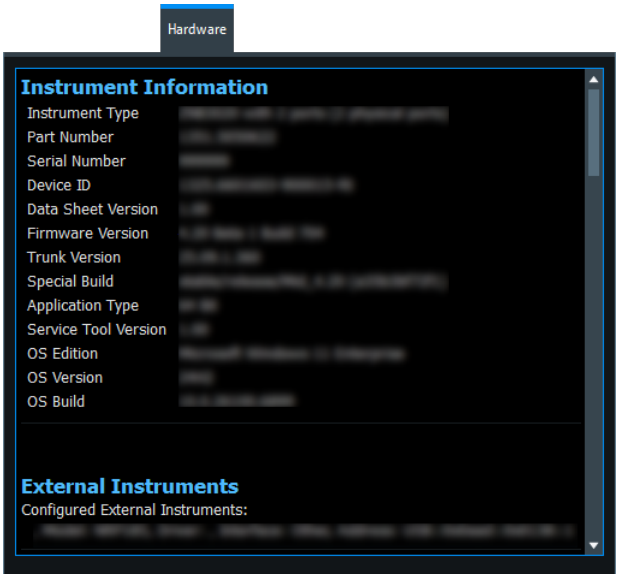


Software Option Info
Software options are listed with their name and description, the option key and key type, and the activation and expiration date (if applicable).

Remote command:
`DIAGnostic:PRODuct:OPTion:INFO?`
`*OPT?`

Hardware tab

Gives an overview of the analyzer's hardware configuration and basic hardware-related instrument settings.



Instrument Information
Provides basic information on the instrument, its firmware and operating system.

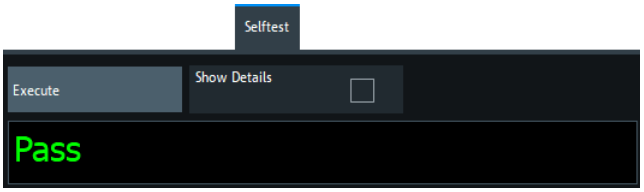
Remote command:
DIAGnostic:SERVice:HWINfo?
DIAGnostic:SERVice:SWINfo?

Configured External Instruments
The "Hardware" tab also provides information about connected external instruments such as power meters or switch matrices. In particular, for mechanical switch matrices the current relay switch counts are reported (if supported by the matrix).

Remote command:
SYSTem:COMMunicate:RDEVice:SMATrix<Matr>:RELays:SWITch:COUNt?

Selftest tab

Allows you to run an automatic self-test on the analyzer, and to display its results. Enable "Show Details" to see the detailed test results.



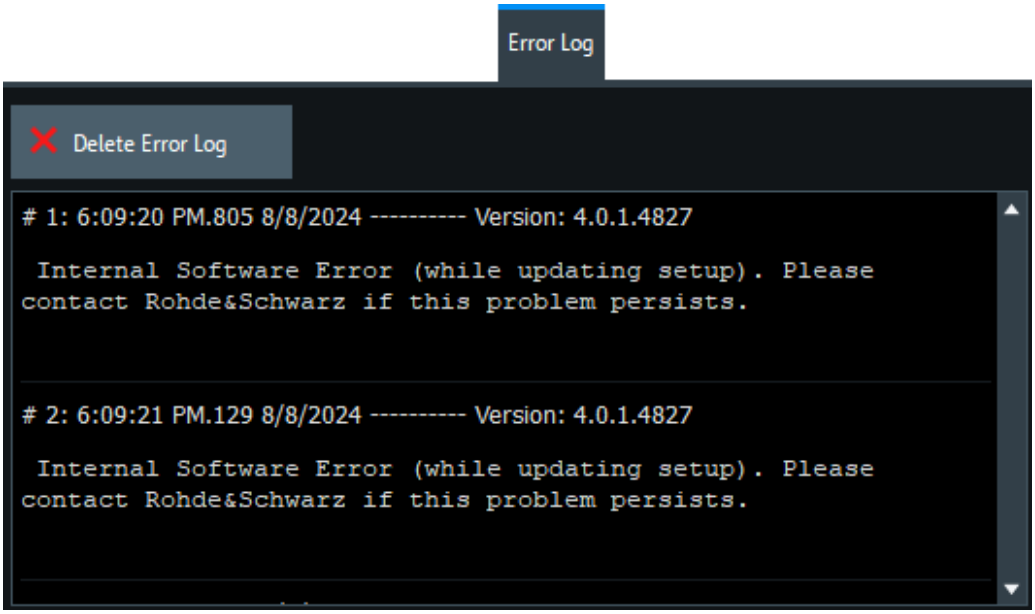
Execute
Runs an automatic self-test.

Remote command:
*TST?

Error Log tab

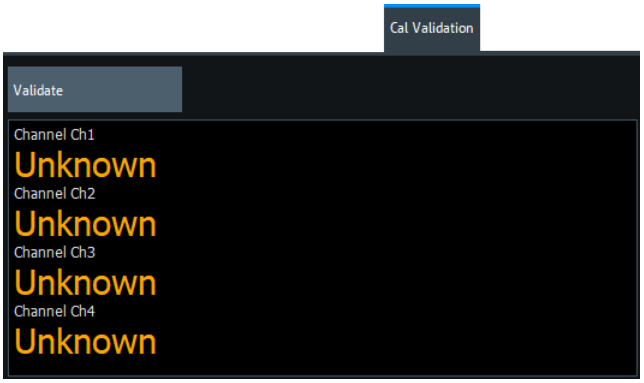
Contains a chronological record of errors that occurred in the current and in previous sessions (see Section 9.1, "Errors during firmware operation", on page 1687).

The "Error" tab comes with an additional button for clearing the log ("Delete Error Log").



Cal Validation tab

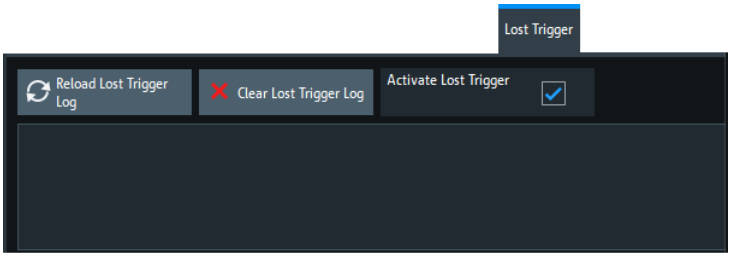
Displays the result of the last cal validation. See "Cal Validation ..." on page 564.



Lost Trigger tab

Displays the lost trigger events that occurred in the current and in previous VNA sessions ("lost trigger event log").

For each sweep with at least one lost trigger event, a log entry is created. The timestamp of an entry indicates the end time of the related sweep.



Reload Lost Trigger Log
While the "Lost Trigger" tab is open, the display is not refreshed automatically. Use "Reload Lost Trigger Log" to reload the lost trigger log file.

Remote command:
`TRIGger[:SEQuence]:LTRigger:COUNt?`

Clear Lost Trigger Log
Does what it says. Also resets the lost trigger counter.

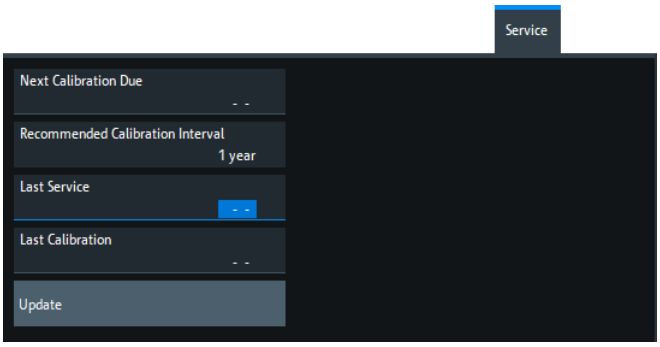
Remote command:
`TRIGger[:SEQuence]:LTRigger:RESet`
`TRIGger[:SEQuence]:LTRigger:COUNt?`

Activate Lost Trigger
Activates/deactivates tracking of lost trigger events.

Remote command:
`TRIGger[:SEQuence]:LTRigger[:STATe]`

Service tab

Allows you to read or specify service-related information. This information can be read via R&S HUMS, however, it does not require the corresponding option R&S ZNB-K980.



Next Calibration Due
The date (and time) the instrument needs calibration to be done.

Remote command:
`DIAGnostic:SERVice:CALibration:DUE:DATE`

Recommended Calibration Interval
Queries the recommended calibration interval, set by Rohde & Schwarz service.

Remote command:
`DIAGnostic:SERVice:CALibration:INTerval?`

Last Service
The last date (and time) the instrument was serviced.

Typically set by Rohde & Schwarz service.

Remote command:
`DIAGnostic:SERVice:DATE`

Last Calibration
The last date (and time) the instrument was calibrated.

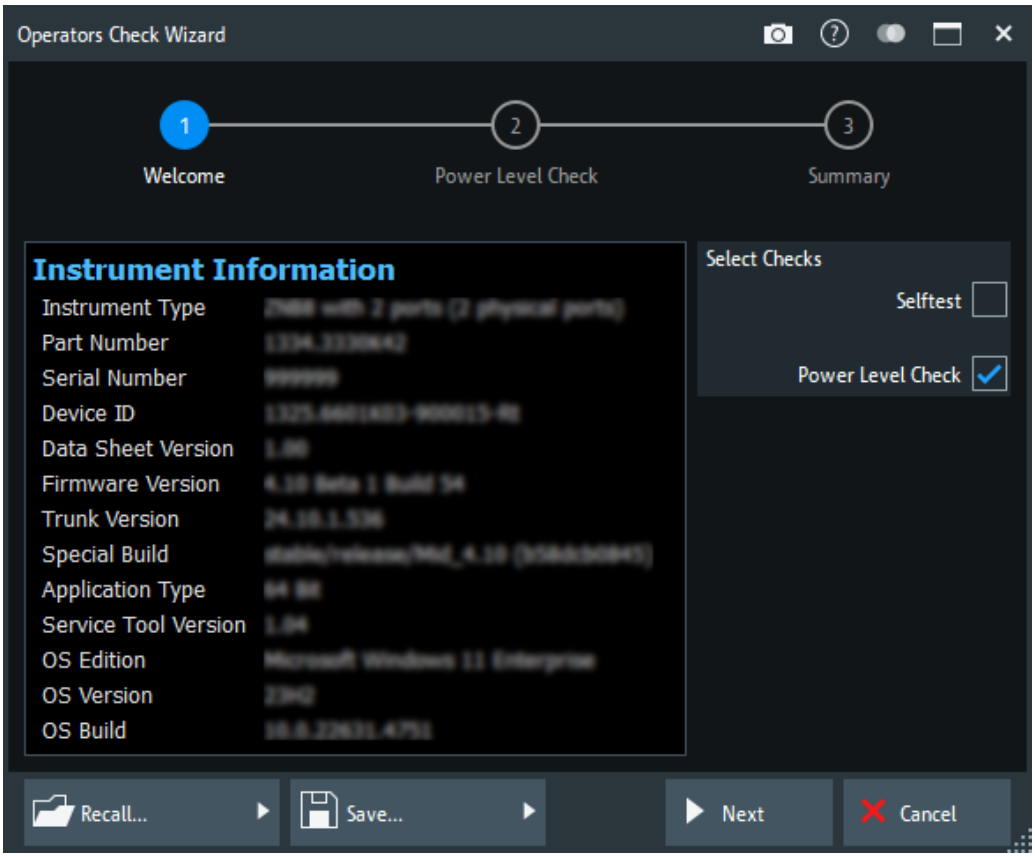
Typically set by Rohde & Schwarz service.

Remote command:
`DIAGnostic:SERVice:CALibration:DATE`

5.17.1.4 Operators Check wizard

Welcome page

The "Operators Check" wizard allows you to perform self-tests and other helpful checks from one central place in the analyzer GUI. The start page of the wizard is the selection page for the checks, its final page is the summary of all selected and executed checks.



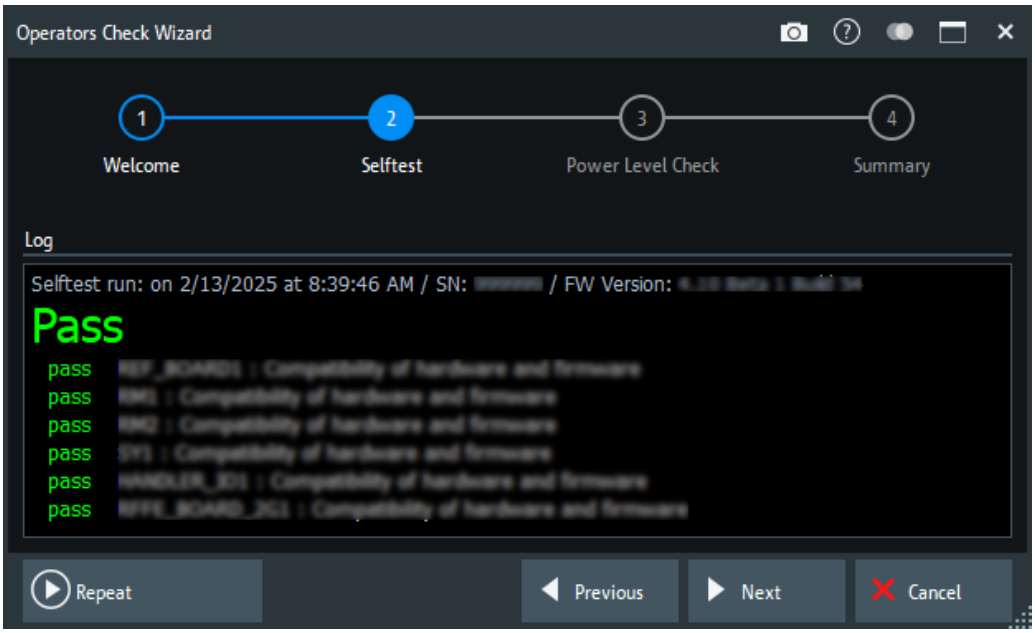
Recall
Allows you to load an operators check definition from an existing "OperatorsCheck XML" (`*.opch`) file. Such a file can be created with the Save function of the "Operators Check" wizard.

Save
Saves the current "Operators Check" definition to a "OperatorsCheck XML" (`*.opch`) file.

Note that if the "Operators Check" definition uses custom settings, then the `opch` file possibly references external files. E.g., for power checks with custom limit lines, the generated "OperatorsCheck XML" file `<filename>.opch` references external limit line files `<filename>_opch<i>.line`. To be able to recall the operator check definition, keep all these files in one directory and do not rename the external files.

Selftest page

In case you selected "Selftest" on the Welcome page, the next step in the "Operators Check" wizard is the standard selftest functionality of the R&S ZNB firmware.



Execute/Repeat
Executes the self-test for the first time in this "Operators Check" wizard session, or repeats it.

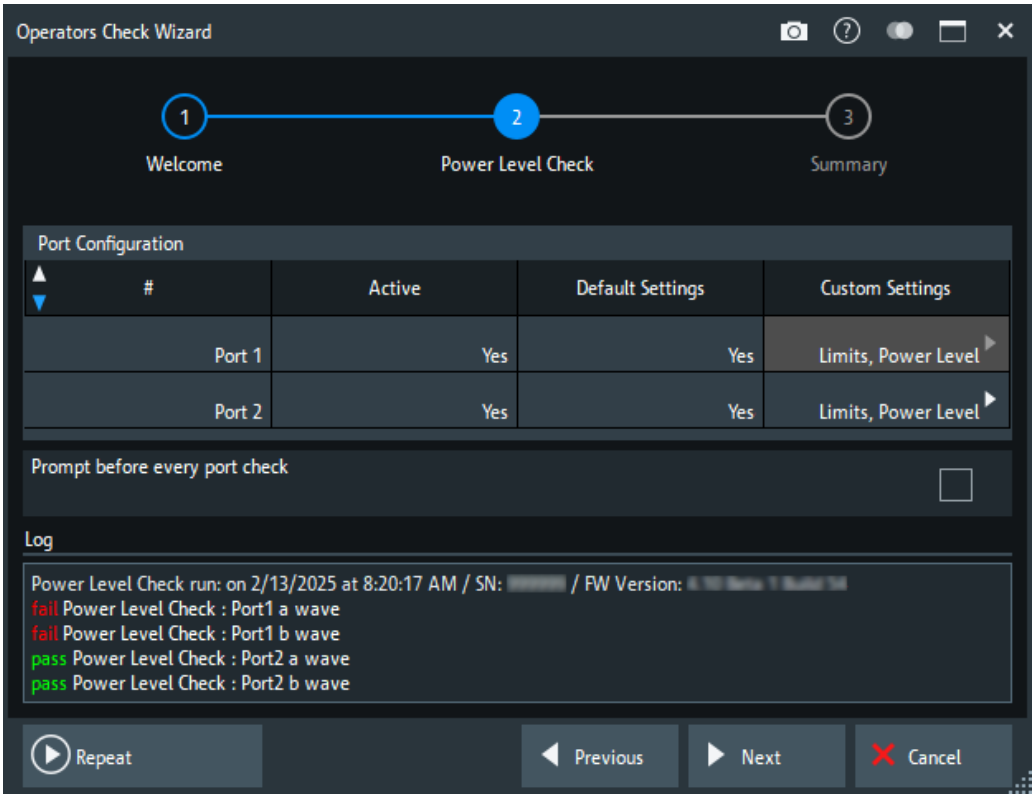
Remote command:
`SYSTem:OPCHeck:SELFtest?`

Power Check page

In case you selected "Power Check" on the Welcome page, you can configure and run port power checks from this page.

During the port power check, the firmware performs a 201-point-linear frequency sweep. By default:

- The sweep covers the whole frequency range of the analyzer.
- The source power is set to -10 dBm and the firmware checks whether the measured a- and b-waves are at most 3 dBm above or below this value.



The upper part of the page allows you to define the checks, which can then be run with the "Execute" button. The results are displayed in the "Log" area.

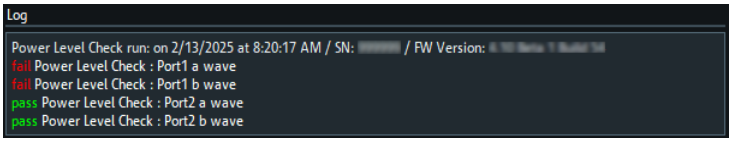


Figure 5-97: Log area

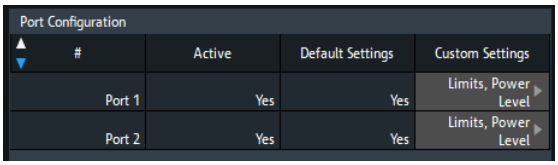
The power check is not a complete verification of the instrument's performance. It is for indication only and shows that the source and receivers of the R&S ZNB are functioning correctly.

Under some circumstances, the default limits are not met. Typically, small limit violations mean that the default factory calibration data has drifted slightly from last time it was sent for a manufacturer's calibration and **not** that the instrument is faulty. Small deviations don't impact measurement validity and accuracy, once you have performed your own user calibration.

The power check can also give varying results if the test ports are left open, rather than connecting an Open from a calibration kit. To be sure, activate Prompt before every port check and connect an Open or Short during the check instead of leaving the ports open.

Power check definition

For each physical VNA port, you can decide whether to perform a power check ("Active"), whether to use default or custom settings ("Default Settings"), and whether you want to ask the firmware to "Prompt before every port check".



Active
Defines whether a power test shall be executed for the related port.

Default Settings
Defines whether default or custom power and limit line settings shall be used for the power test of the related port.

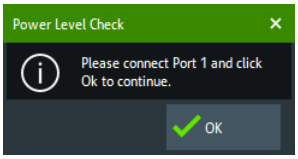
The control is only enabled, if this port is set to Active. If you disable it, a tap/click on the respective Custom Settings cell allows you to customize the port power test.

Custom Settings
If Default Settings is disabled, a click/tap on the respective "Custom Settings" field opens the "Define Limit Lines and Power Level" dialog. This dialog allows you to set the source power for the test of the respective port, and to define port-specific limit lines for the test.

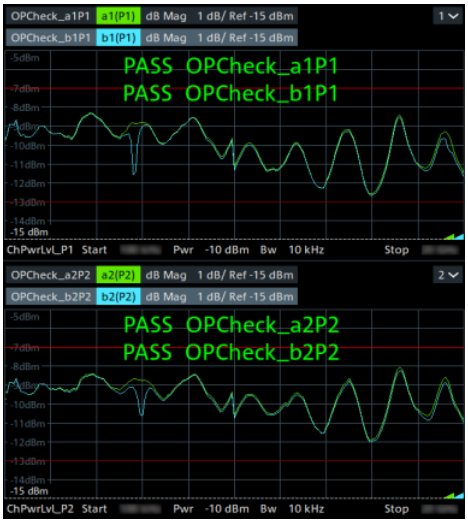


The available GUI elements for the limit line definition are the same as in the general-purpose limit line setup dialog. For their description, see Section 5.6.1.2, "Define Limit Lines dialog", on page 488.

Prompt before every port check
This option comes in handy if you want to connect a particular device such as a reflection standard to the related port before you execute or repeat the test.



Execute/Repeat
Executes or repeats the "Active" port power tests with their current limit line and power level settings. The firmware displays the relevant a- and b-wave traces, limit lines and PASS/FAIL results and in the diagram area:

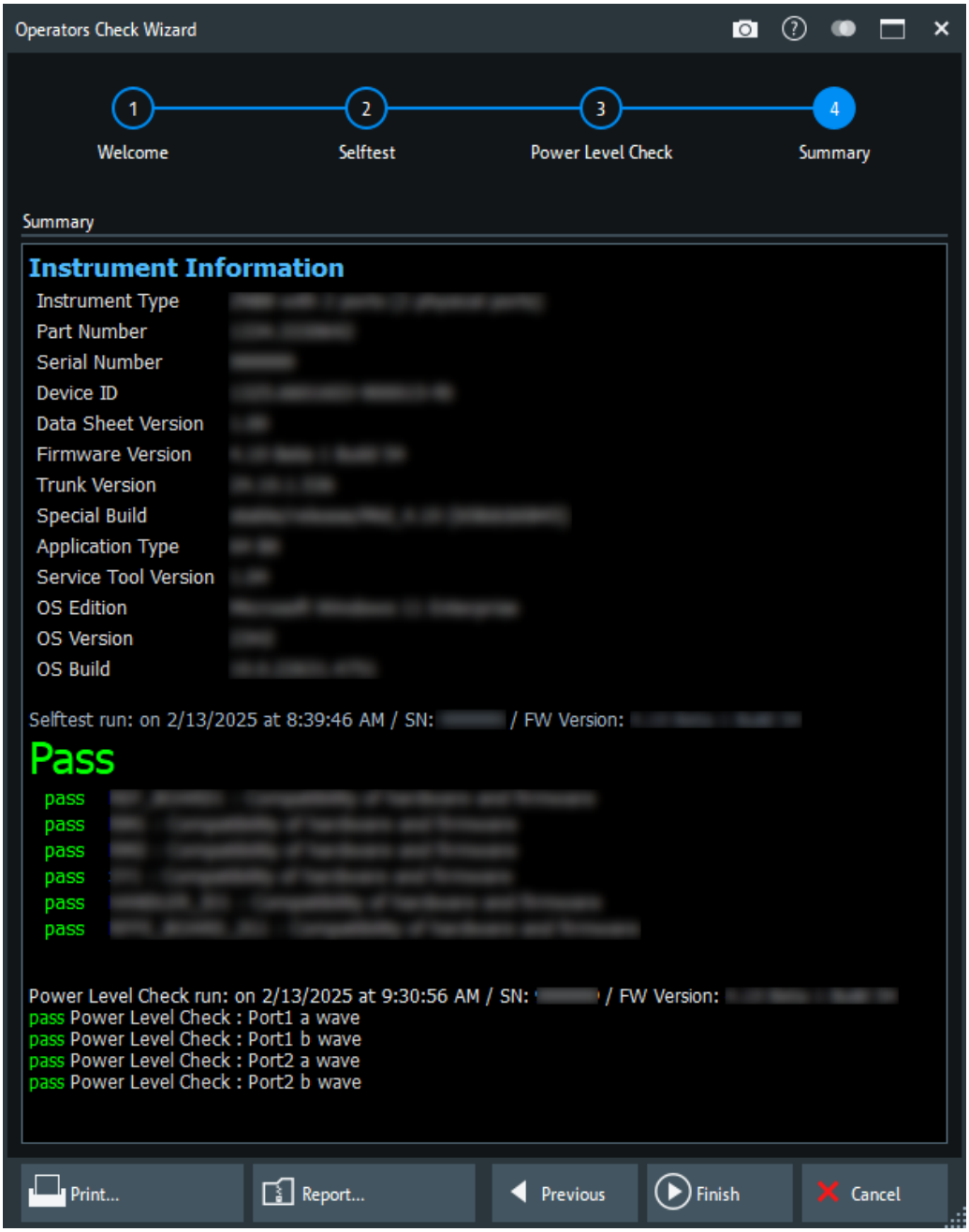


The PASS/FAIL results are also written to the Log area.

Remote command:
`SYSTem:OPCHeck:PLEVel:PORT<Port>?`

Summary page

Summarizes the check results and allows you to print them.



Print
Allows you to print the "Summary" of the executed tests, similar to the Print... functionality of the system "Info" dialog.

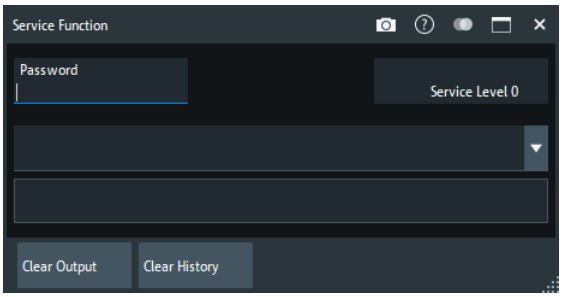
Report
Same functionality as Create R&S Support Information in the system "Info" dialog.

5.17.1.5 Service Function dialog

The "Service Function" dialog gives access to the service functions of the instrument.

Access: [Setup] > "Setup" > "Service Function"

Most of the service functions require a service level > 0 that is protected by a password. Those service functions should be used by a Rohde & Schwarz service representative only. Refer to the service manual for more information.



Password
Enter a password here to activate the required service level.

Remote command:
`SYSTem:PASSword[:CENable]`

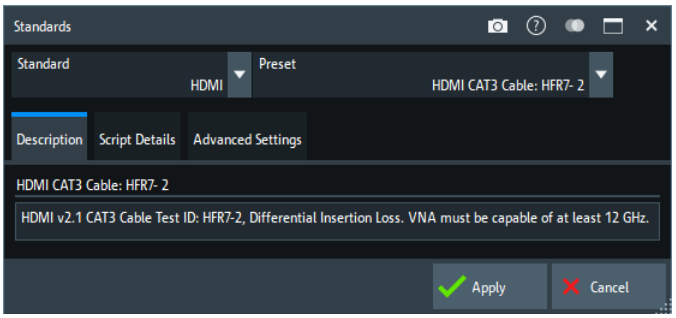
Service Function
Identifier of the service function in "dotted textual" (example: sw.common.memory_usage) or "dotted decimal" (example: 0.1.18.0) representation.

Remote command:
`DIAGnostic:SERVice:SFUNction`

5.17.1.6 Standards dialog

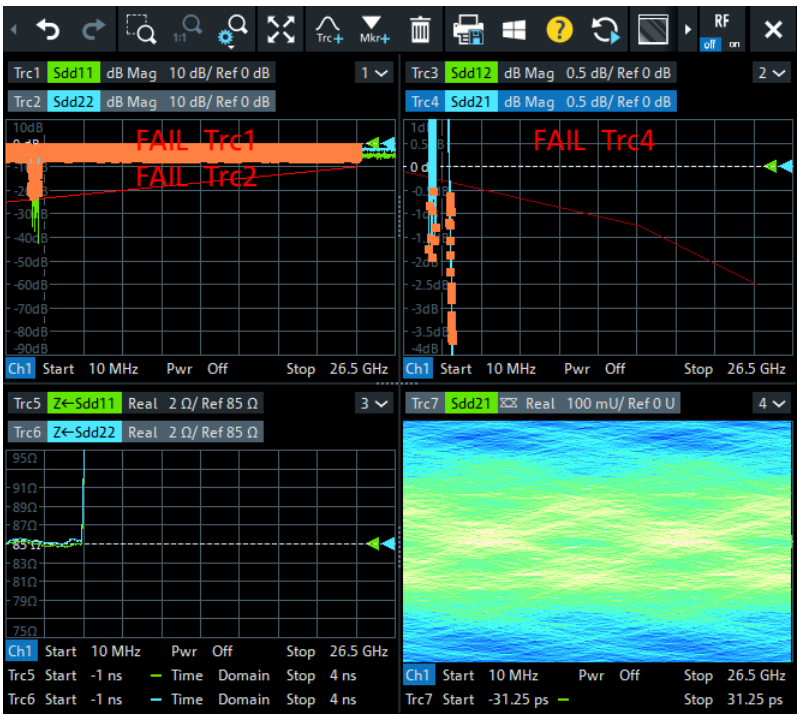
Allows you to set up your instrument for measuring DUTs of various industry standards.

The setups are based on predefined SCPI-based "Preset" scripts with IECWin special commands, which enable conditional script execution based on the properties of your instrument (frequency range, number of ports, available options, ...).



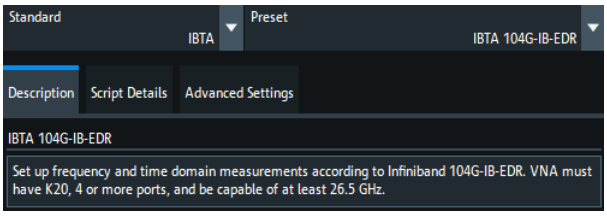
Access: Standards tab > "Standards"

If you execute the "PCIe 5 Connector" script, for example, the result looks something like this:

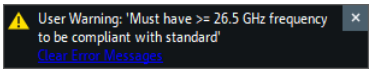


Description tab

Provides a description of the selected "Preset" script, specifying the requirements for successful execution (number of ports, frequency range, available options, ...).

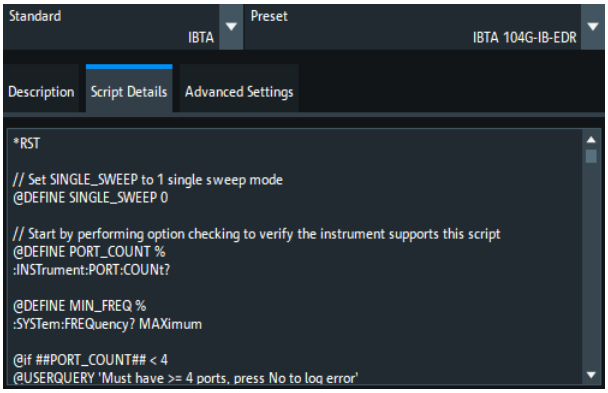


The predefined scripts do not stop on minor errors, such as an insufficient maximum frequency. However, in this case they tell you that the resulting setup and measurements do not meet the requirements of the underlying standard conformance tests.



Script Details tab

Mirrors the SCPI/IECWin commands to be executed for the selected "Preset".

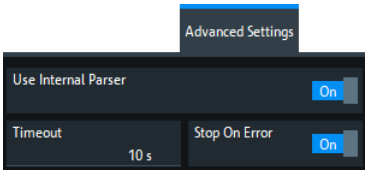


In the screenshot above, you can see the script using IECWin variable definitions (`@DEFINE ...`), conditional command execution (@if <condition> ... @else ...) and user interaction (`@USERQUERY ...`).

Advanced Settings tab

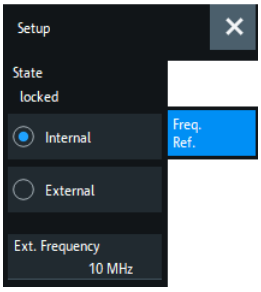
Allows you to configure how the script is executed:

- Use the internal parser (default) or IECWin (a.k.a. "GPIB Explorer")
- Timeout
- Stop or continue on error



5.17.2 Freq. Ref. tab

Selects a reference signal for synchronization between the R&S ZNB and external devices. A common reference frequency is advisable to ensure frequency accuracy and frequency stability in the test setup.



State
Indicates the state of the internal phase locked loop: If the frequencies are properly synchronized, the state is "locked".

Remote command:
n/a

Internal/External
Selects the internal or an external reference clock signal for synchronization.

- **Internal:** The analyzer synchronizes to its internal 10 MHz reference clock.
  Use the REF OUT connector at the rear of the instrument to synchronize other devices (e.g. signal generators or a second R&S ZNB network analyzer).
- **External:** The analyzer synchronizes to an external reference clock via the REF IN connector at the rear of the instrument.
  The external reference signal has to meet the specifications of the data sheet; its frequency must be specified in the "Ext Frequency" field. The internal reference signal is synchronized to the external signal. The external signal is also looped to REF OUT, so that it can be reused to synchronize other devices.

Remote command:
`[SENSe:]ROSCillator[:SOURce]`

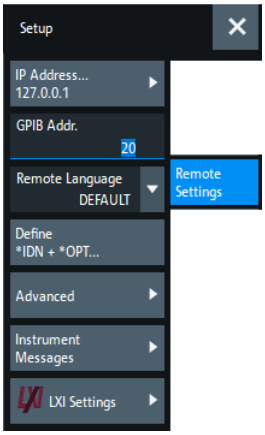
Ext Frequency
Specifies the frequency of the external reference clock signal at REF IN.

Remote command:
`[SENSe:]ROSCillator:EXTernal:FREQuency`

5.17.3 Remote Settings tab

Configures the remote control interfaces of the R&S ZNB.

5.17.3.1 Controls on the Remote Settings tab



IP Address
Displays the current IP4 address of the R&S ZNB. By default, the analyzer is configured to use dynamic TCP/IP configuration (DHCP) and obtain all IP address information automatically. See Section 3.1.12.1, "Assigning an IP address", on page 31.

The button opens the Windows "Network Connections" system config dialog that allows you to configure the IP4 (and IP6) settings of your instrument.

GPIB Address
Defines the analyzer's GPIB address. The address must be in the range between 0 and 30.

Remote command:
`SYSTem:COMMunicate:GPIB[:SELF]:ADDRess`
`SYSTem:COMMunicate:GPIB[:SELF]:DCLear:SUPPress`

Remote Language
Selects the syntax of the R&S ZNB's instrument control commands.

- The DEFAULT language corresponds to the commands reported in this documentation; see Section 7.3, "SCPI command reference", on page 917. The byte order for binary data is set to little endian (see `FORMat:BORDer SWAPped`).
- The ZVABT language ensures compatibility with network analyzers of the R&S ZVA/B/T family. E.g., compared to the DEFAULT language, the command set does not include `INITiate:CONTinuous:ALL` and `INITiate[:IMMediate]:ALL`, and the function of `INITiate:CONTinuous` and `INITiate[:IMMediate][:DUMMy]` is modified. Refer to the remote control documentation in Section 7.3.8, "INITiate commands", on page 1247. The byte order for binary data is set to little endian.
- The ZVR language ensures compatibility with network analyzers of the R&S ZVR family. See also Section 7.5, "R&S ZVR/ZVABT compatible commands", on page 1620. The byte order for binary data is set to little endian.
- The other languages define command sets for network analyzers from other instruments or manufacturers. The byte order for binary data is set to big endian (see `FORMat:BORDer NORMal`).

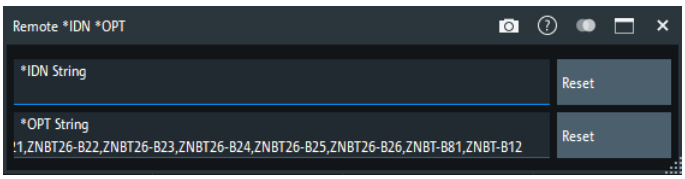
Note: Remote languages other than DEFAULT are intended for remote control only. A mixed approach, with parts of the instrument configuration defined via the GUI can cause unexpected results.

Remote command:
`SYSTem:LANGuage`

Define *IDN + *OPT...
Opens the "Remote *IDN + *OPT..." dialog, which allows you to define the ID string (`*IDN?`) and the option string (`*OPT?`) of the analyzer.

- If the DEFAULT language is activated, the factory ID and option strings are used. The factory ID contains information on the instrument and firmware. Its format can be changed using `SYSTem:FORMat:IDENtify`. The option string is a comma-separated list of all installed software and hardware options.
- The default strings are automatically adjusted to the selected Remote Language.
  - If the PNA language is activated, Keysight-compatible ID and option strings are set. The bit order for transferred binary data is normal.
  - If one of the "HP xxxx" languages is activated, "HP xxxx"-compatible ID and option strings are set. Binary data is transferred in a device-specific bit order, however, the bit order can be changed using "HP xxxx"-specific commands.

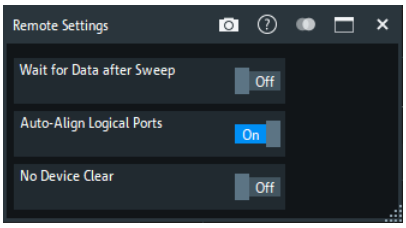
The ID and option strings can be set manually, or reset to the factory defaults.



Remote command:
SYSTem:IDENtify
SYSTem:IDENtify:FACTory
SYSTem:FORMat:IDENtify
SYSTem:OPTions[:STRing]
SYSTem:OPTions:FACTory
FORMat:BORDer

Advanced ...
Opens a dialog that allows to set the following parameters:

- "Wait for Data after Sweep" determines the execution behavior of `INITiate[:IMMediate]` commands (see Section 7.3.8, "INITiate commands", on page 1247).
  If enabled, an automatic `*WAI` is added (see Section 7.2, "Common commands", on page 913). By default, it is disabled.
- "Auto-Align Logical Ports" determines the logical port creation logic.
  If enabled (default), logical ports are aligned and must be set from low to high port (which was the only possibility before firmware V1.91). If set to disabled, new ports can be created freely, like in manual operation.
- "No Device Clear" suppresses Device Clear (DCL, SDC) GPIB interface messages.



Remote command:
SYSTem:COMMunicate:GPIB[:SELF]:INIT:WAIT
SYSTem:COMMunicate:GPIB[:SELF]:LPORt:ALIGn
SYSTem:COMMunicate:GPIB[:SELF]:DCLear:SUPPress

Instrument Messages...
Opens the Advanced tab of the "System Config" dialog. In the "Instrument Messages" on page 820 section you can configure the display of instrument messages as information popups, in particular the display of remote control command errors.

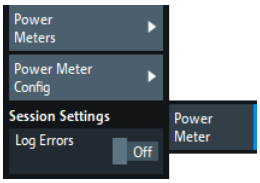
5.17.4 Power Meter tab

Allows you to set up and configure power meters and to enable error logging for the current session.

Remote control via GPIB

To control external devices via GPIB, you have to use an Section 4.7.27, "USB-to-IEC/IEEE adapter", on page 267.

5.17.4.1 Controls on the Power Meter tab



The buttons in the "Power Meter" tab open the following dialogs:

- "Power Meters", see Section 5.17.4.2, "External Power Meters dialog", on page 845
- "Power Meter Config", see Section 5.17.4.3, "External Power Meter Config dialog", on page 849
  This button is active if at least one external power meter R&S NRP is online (physically connected, switched on, ready to be used).

Log Errors
Enables the transfer of error messages for external devices (e.g. connection errors) to the error log. The error log appears in the "Info" dialog; see Section 5.17.1.3, "Info dialog", on page 826.

Remote command:
n/a

5.17.4.2 External Power Meters dialog

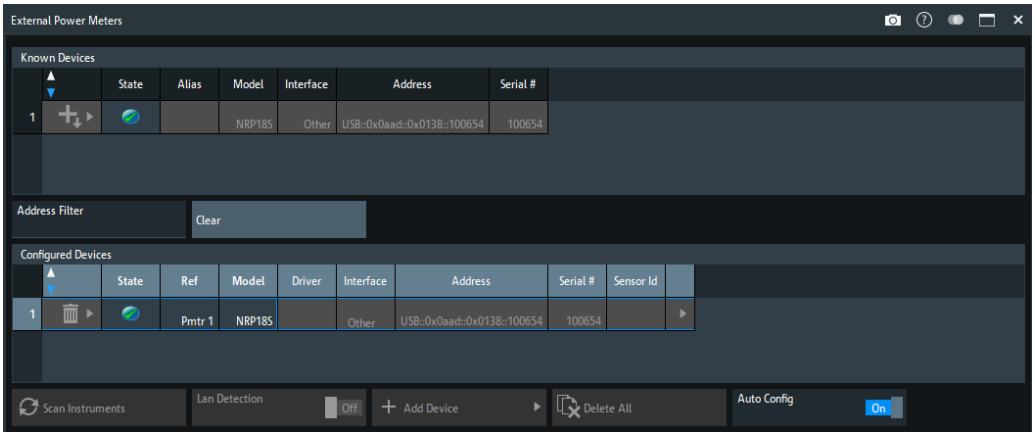
The "External Power Meters" dialog configures external power meters so that they can be used for measurements and power calibrations.

Access: System – [Setup] > "External Devices" > "Power Meters..."

Access: [Cal] > "Power Cal Settings" > "Power Meters..."

Background information

Refer to section Section 4.7.28, "External power meters", on page 267.



Remote control via GPIB

- To control external devices via GPIB, you have to use an Section 4.7.27, "USB-to-IEC/IEEE adapter", on page 267.
- To control external devices via GPIB, the pre-installed RS Visa library (`visa32.dll`) must be replaced by the **NI Visa** library.

The configuration of a new external power meter involves the following steps:

1. Connect the power meter to your VNA using a LAN, GPIB, or USB interface.
2. If the power meter is connected via LAN, enable LAN Detection
3. Select Scan Instruments and wait until the power meter appears in the table of "Known Devices".
4. Select to add the power meter to the list of Configured Devices.

If your VNA fails to detect a connected power meter:

► Select Add Device to define the interface type and address.

If successful, your VNA can auto-detect the instrument type (driver) and the serial number of the connected power meter.

Known Devices
Table with all power sensors/meters the analyzer detects to be on line (i.e. connected and switched on). "Scan Instruments" refreshes the table.

Select to add a power sensor/meter to the table of Configured Devices.

Note: Even though an R&S NRP2 power meter can appear in the table of "Known Devices", adding it to the list of "Configured Devices" directly results in an invalid configuration. Use Add Device to configure it manually instead.

Remote command:
n/a

Configured Devices
Table with all power meters in use with their properties.

Except for the auto-detected Known Devices, to appear in the table of "Configured Devices" a power sensor/meter must have been configured manually using Add Device. The properties of manually configured power meters (Add Device) can be changed in the table cells.

The **"Sensor Id"** column is used for power meters that can control several power sensors, such as the R&S NRP2. By selecting the suitable "Sensor Id", you can address the related power sensor.

The R&S NRP2, for example, has four power sensor connectors, Sensor A to Sensor D. By selecting "A", "B", "C", or "D" you can address the power sensor that is attached to this connector.

Use the icon beneath the "Sensor Id" cell to take control over another power sensor attached to the same power meter.

The following symbols (grayed out for *used* devices) indicate the status of the respective device:

- – The device is online (connected, switched on, ready to be used).
- – There is a problem with the device.
  This state can be caused by different problems:
  – General communication error
    In this case, check whether the device is properly connected to the configured interface.
  – Self test error
    In this case enable error logging for external devices (see "Log Errors" on page 845), and search the Error Log tab for self test error codes of the device.
- – The device driver does not match exactly the device type. Probably the device cannot be controlled properly. Measurements results are questionable.

Remote command:

```
SYSTem:COMMunicate:RDEVice:PMETer<Pmtr>:DEFine
SYSTem:COMMunicate:RDEVice:PMETer:DELete
SYSTem:COMMunicate:RDEVice:PMETer:CATalog?
SYSTem:COMMunicate:RDEVice:PMETer:COUNt?
```

Scan Instruments
Performs an autodetection of "Known Devices" on the supported interfaces.

Note: Unintentional switchover to remote control

When using the NI-VISA library, ensure that the network analyzer itself is not listed as a network device in the Measurement & Automation Explorer. Otherwise, "Scan Instruments" initiates an identification query (`*IDN?`), causing the analyzer to close the "External Power Meters" dialog (without executing "Scan Instruments") and to activate the remote screen.

Remote command:

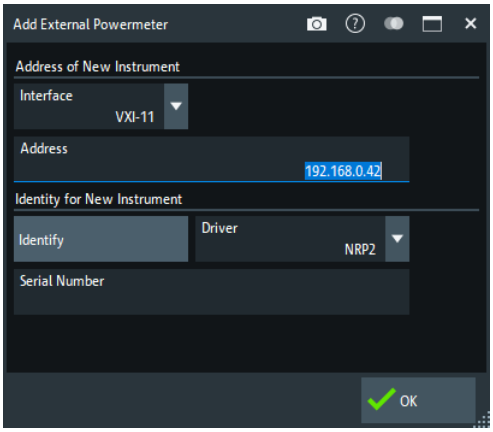
```
SYSTem:COMMunicate:RDEVice:PMETer:SCAN?
```

LAN Detection
Activate "LAN Detection" to include the LAN interface in the autodetection sequence (see Scan Instruments).

Note: "LAN detection" only works for external devices on the same IP subnet than your VNA.
As a prerequisite, your VNA must accept incoming connections on UDP port 2473. If necessary, add a corresponding inbound rule to your firewall settings.

Remote command:
n/a

Add Device
Adds a new instrument to the list of "Configured Devices".



In the "Add External Power Meter" dialog, you can specify the instrument and connection properties:

- "Interface" selects an interface/protocol type for the connection. In addition to the HiSLIP, VXI-11, SOCKET, GPIB, and "Raw Serial" interface types, the analyzer accepts any "Other" interface supported by the installed VISA library.
  - HiSLIP, VXI-11 and SOCKET is applicable for devices connected to the LAN connector on the rear panel of the analyzer.
  - GPIB is applicable for devices that shall be controlled via GPIB.
  - "Other" is used in particular for USB connections, e.g. for auto-detected R&S NRP-Zxx sensors.
- "Address" contains the address for the current interface type. GPIB addresses must be unique for all devices connected to the GPIB bus (range: 0 to 30), GPIB and IP addresses must agree with the entries in the VISA library. The remaining interface types require composite address formats; see Table 5-11.
  If a power meter is connected to your VNA, the entries in the [Driver for New Instrument] panel can be auto-detected for the specified interface type and address.
- "Identify" sends an identification query ("IDN?") to the specified device address to identify the type and serial number of the connected power meter and select an appropriate driver file. Power meter driver files (`*.pwm`) are stored in the `Resources\ExtDev` subdirectory of the analyzer's program directory.

Table 5-11: Interface types for external power meters and address formats

Physical interface (connector)	Interface (protocol)	Address	Remarks
LAN	HiSLIP	<IP Address> e.g. 192.168.0.42	Full VISA resource string: TCPIP[board]::<IP Address> ::hislip0[::INSTR]
	VXI-11		Full VISA resource string: TCPIP[board]::<IP Address> [::INSTR]
	SOCKET	<IP Address>::<PortNo> e.g. 192.168.0.77::50000	LAN connection with pure TCP/IP protocol TCPIP[board]::<IP Address> ::<Port No>::SOCKET
GPIB	GPIB0 ... GPIB9	<GPIB Address> e.g. 20	Full VISA resource string: GPIB[board]::<Address>[::INSTR]
LAN or USB	Other	Interface-specific For USB-VISA: USB::<ManID>::<ProdID>::<SerialNo>::INSTR e.g. USB::0x0AAD::0x0138::100654, where: • 0x0AAD (2733) is the manufacturer ID of Rohde & Schwarz. • 0x0138 (312) is an example for a R&S product ID (R&S NRP18S). • The serial number is device-specific.	Use full VISA resource string; refer to your VISA user documentation.

Remote command:
SYSTem:COMMunicate:RDEVice:PMETer<Pmtr>:DEFine

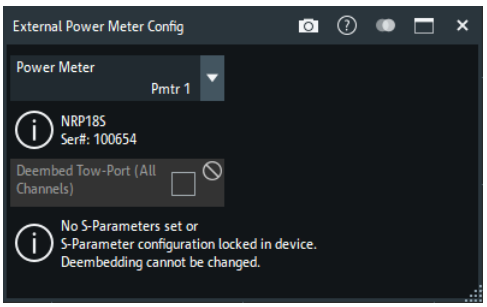
Auto Config
Causes the analyzer to clear the lists of "Known Devices" and "Configured Devices" and to configure all R&S NRPxxS/SN power sensors, detected at any of the USB ports, automatically. No manual configuration is required.

Remote command:
SYSTem:COMMunicate:RDEVice:PMETer:CONFigure:AUTO[:STATe]

5.17.4.3 External Power Meter Config dialog

This dialog provides information on the connected power meters. It is available for Rohde & Schwarz power sensors R&S NRP only.

Access: Power Meter tab > "Power Meter Config"



Power Meter
Select one of the connected R&S NRP power sensors to read (and change) its properties.

Deembed Two-Port (All Channels)
Power sensors R&S NRP-Z can use an `s2p` Touchstone file stored on the device to deembed a two-port network attached to its RF port. If such a (user-defined) file is available on the sensor, you can enable/disable "Deembed Two-Port (All Channels)" to activate/deactivate this deembedding on the sensor persistently.

See application note 1GP70 "Using S-Parameters with R&S®NRP-Z Power Sensors" for background information. It is available on the Rohde & Schwarz internet site at https://www.rohde-schwarz.com/appnotes/1GP70.

Remote command:
`SYSTem:COMMunicate:RDEVice:PMETer<Pmtr>:SPCorrection[:STATe]`

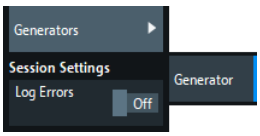
5.17.5 Generator tab

Allows you to set up and configure external generators and to enable error logging for the current session.

Remote control via GPIB

- To control external devices via GPIB, you have to use an Section 4.7.27, "USB-to-IEC/IEEE adapter", on page 267.
- To control external devices via GPIB, the pre-installed RS Visa library (`visa32.dll`) must be replaced by the **NI Visa** library.

5.17.5.1 Controls on the Generator tab



"Generators..." opens the "External Generators" Dialog; see Section 5.17.5.2, "External Generators dialog", on page 851.

Log Errors
Enables the transfer of error messages for external devices (e.g. connection errors) to the error log. The error log appears in the "Info" dialog; see Section 5.17.1.3, "Info dialog", on page 826.

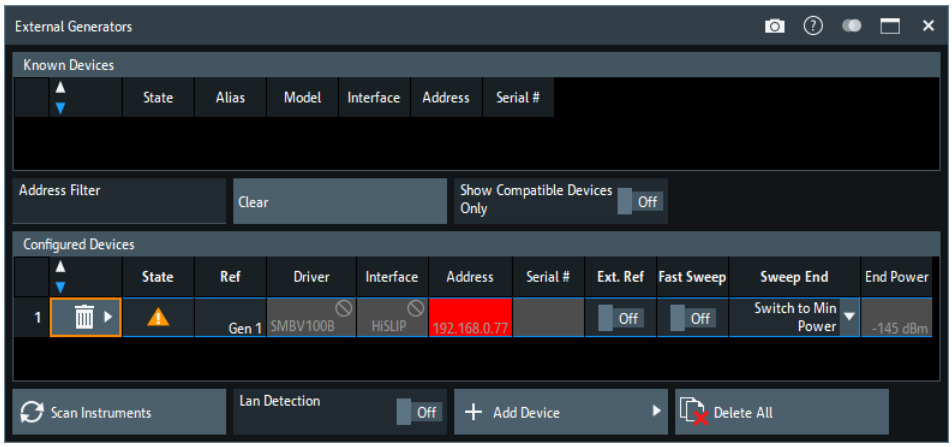
Remote command:
n/a

5.17.5.2 External Generators dialog

The "External Generators" dialog configures external generators so that they can be used for measurements and power calibrations.

Access: System – [Setup] > "External Devices" > "Generators..."

Background information
Refer to section Section 4.7.29, "External generators", on page 269.



The configuration of a new external generator involves the following steps:

1. Connect the generator to your R&S ZNB using a LAN (HiSLIP, VXI-11, SOCKET), GPIB, or USB interface.
2. If the generator is connected via LAN, enable "LAN detection"
3. Click "Scan Instruments" and wait until the generator appears in the table of "Known Devices".
4. Click + to copy the generator into the list of configured devices.

If the R&S ZNB fails to detect a connected generator,

► Click "Add Device" to define the interface type and address.

The R&S ZNB can auto-detect the instrument type (driver) and the serial number of the connected generator.

To control external devices via GPIB, the pre-installed RS Visa library (`visa32.dll`) must be replaced by the **NI Visa** library.

Known Devices
Table with all generators that the analyzer detects to be on line (i.e. connected and switched on). "Scan Instruments" refreshes the table; copies a detected instrument to the table of "Configured Devices".

Remote command:
n/a

Show Compatible Devices
If enabled, only supported generators are displayed in the list of Known Devices.

Remote command:
n.a.

Configured Devices
Table with all generators in use with their properties. The properties of manually configured generators ("Add Device", opens the "Add External Generator" dialog) may be changed in the dialog.

The following generator properties are not defined in the "Add External Generator" dialog but configurable here:

- "Ext. Ref." switches the generator to either external (control enabled) or internal frequency reference. The setting does not affect the frequency reference settings of the analyzer System – [Setup] > "Freq. Ref.". Make sure to establish consistent settings in your test setup (one instrument is the master, the others use external reference frequency). See "Reference frequency" on page 270.
- "Fast Sweep" enables or disables the fast sweep mode for external generators that support a frequency and level list mode (triggered mode). See "Fast sweep mode and conditions" on page 271.
- "Sweep End" defines the output power of the external generator after the end of a sweep or sweep sequence (single sweep mode; Channel – [Sweep] > "Sweep Control" > "Single"). The minimum power value ("Switch to Min. Power") depends on the generator type. "Set to Selected Power" means that the generator is commanded to transmit at the selected "End Power".

The following symbols (grayed out for *used* devices) indicate the status of the respective device:

- – The device is online (connected, switched on, ready to be used).
- – There is a problem with the device.
  This state can be caused by different problems:
  – General communication error
    In this case, check whether the device is properly connected to the configured interface.
  – Self test error
    In this case enable error logging for external devices (see "Log Errors" on page 845), and search the Error Log tab for self test error codes of the device.
- – The device driver does not match exactly the device type. Probably the device cannot be controlled properly. Measurements results are questionable.

Remote command:
SYSTem:COMMunicate:RDEVice:GENerator<Gen>:DEFine
SYSTem:COMMunicate:RDEVice:GENerator:DELete
SYSTem:COMMunicate:RDEVice:GENerator<Gen>:CATalog?
SYSTem:COMMunicate:RDEVice:GENerator<Gen>:DEFine
SYSTem:COMMunicate:RDEVice:GENerator<Gen>:SEPMode
SYSTem:COMMunicate:RDEVice:GENerator<Gen>:SEPower

Scan Instruments
Performs an autodetection of "Known Devices" on the supported interfaces.

Note: Unintentional switchover to remote control

When using the NI-VISA library, ensure that the network analyzer itself is not listed as a network device in the Measurement & Automation Explorer. Otherwise, "Scan Instruments" initiates an identification query (`*IDN?`), causing the analyzer to close the "External Power Meters" dialog (without executing "Scan Instruments") and to activate the remote screen.

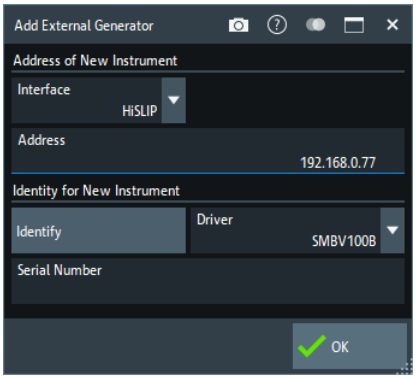
Remote command:
SYSTem:COMMunicate:RDEVice:GENerator:SCAN?

LAN Detection
Activate "LAN Detection" to include the LAN interface in the autodetection sequence (see Scan Instruments).

Note: "LAN detection" will only work for external devices sharing the same IP subnet with the R&S ZNB.
As a prerequisite, the R&S ZNB must accept incoming connections on UDP port 2473. If necessary, add a corresponding inbound rule to your firewall settings.

Remote command:
SYSTem:COMMunicate:RDEVice:GDEVice:LAN:STATe

Add Device
Adds a new instrument to the list of "Configured Devices". In the "Add External Generator" dialog opened, you can specify the properties of the new instrument and the connection.



- "Interface" selects an interface/protocol type for the connection. In addition to the GPIB, VXI-11, HiSLIP, SOCKET, and USB-VISA interface types (for devices con-

nected to the GPIB Bus, LAN or USB connectors of the analyzer; see Table 5-12), the analyzer supports any "Other" interface supported by the installed VISA library.

- "Address" contains the address for the current interface type. GPIB addresses must be unique for all devices connected to the GPIB bus (range: 0 to 30), GPIB and IP addresses must agree with the entries in the VISA library. The remaining interface types require composite address formats; see Table 5-12.
  If an instrument is connected to the R&S ZNB, the entries in the [Driver for New Instrument] panel can be auto-detected for the specified interface type and address.
- "Identify" sends an identification query ("IDN?") to the specified device address in order to identify the type and serial number of the connected generator and select an appropriate driver file. Generator driver files (`*.gen`) are stored in the `Resources\ExtDev` subdirectory of the analyzer's program directory.

Table 5-12: Interface types for external generators and address formats

Physical interface (connector)	Interface (protocol)	Address	Remarks

LAN	HiSLIP	`<IP Address>`
		e.g. `192.168.0.77`
			Full VISA resource string:
			`TCPIP[board]::<IP Address>`
			`::hislip0[::INSTR]`

	VXI-11		Full VISA resource string:
			`TCPIP[board]::<IP Address>`
			`[::INSTR]`

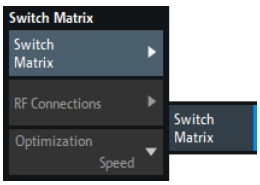
	SOCKET	`<IP Address>::<PortNo>`
		e.g. `192.168.0.77::50000`
			LAN connection with pure TCP/IP protocol
			`TCPIP[board]::<IP Address>`
			`::<Port No>::SOCKET`

GPIB	GPIB0 ... GPIB9	`<GPIB Address>` e.g. `20`	Full VISA resource string: `GPIB[board]::<Address>[::INSTR]`
USB	USB-VISA	`<Man ID>::<Prod ID>::<Ser No>` e.g. `0x0AAD::0x0047::100098`	• `0x0AAD` (2733) is the manufacturer ID of Rohde & Schwarz. • `0x0047` (71) is an example for a R&S product ID (R&S SMF100A). • The serial number is device-specific.
LAN or USB	Other	Interface-specific	Use full VISA resource string; refer to your VISA user documentation.

Remote command:
`SYSTem:COMMunicate:RDEVice:GENerator<Gen>:DEFine`

5.17.6 Switch Matrix tab

Allows you to set up switch matrices.



5.17.6.1 Controls on the Switch Matrix tab

The buttons on the "Switch Matrix" tab open the following dialogs:

- "Switch Matrix...", see Section 5.17.6.2, "External Matrices dialog", on page 855
- "RF Connections...", see Section 5.17.6.4, "Switch Matrix RF Connections dialog", on page 859

"RF Connections" is enabled if at least one switch matrix is configured.

Optimization
Allows you to select between different switch matrix routing optimizations:

- "Speed" – switch as little as possible
- "Precision" – always use the best possible route in terms of quality ("priority")

See Section 4.7.30.4, "Multiple paths: precision vs. speed", on page 275 for background information.

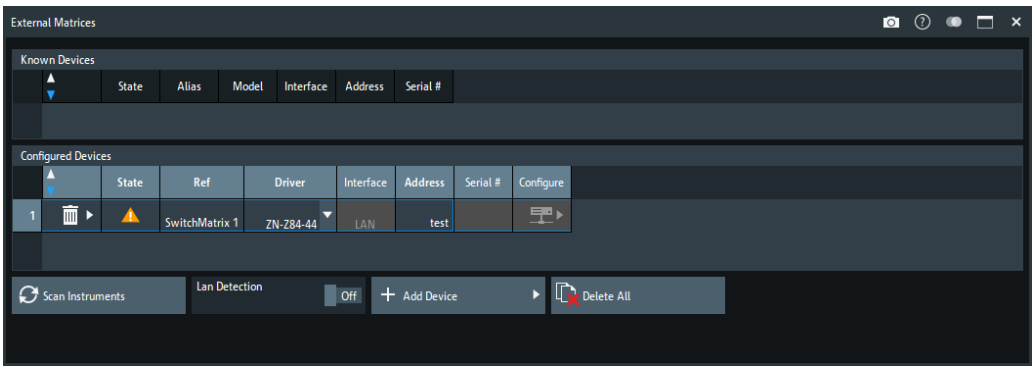
The selection is only enabled, if at least one of the connected switch matrices has multiple routes with non-identical priorities.

5.17.6.2 External Matrices dialog

The "External Matrices" dialog allows you to discover and register external switch matrices.

Access: System – [Setup] > "External Ports" > "Switch Matrix..."

Background information
Refer to section Section 4.7.30, "External switch matrices", on page 271.



The registration of a new external switch matrix typically involves the following steps:

1. Connect the switch matrix to your R&S ZNB via USB or LAN.
2. If you have connected switch matrix and VNA to the LAN:
   a) Enable "LAN Detection".
   b) Select "Scan Instruments".
3. Wait until the switch matrix appears in the table of "Known Devices".
4. Select  to copy it to the list of configured devices.

If "LAN Detection" fails for some reason (see below for details), the external switch matrix can also be registered manually:

1. Select "Add Device" to open the "Add External Switchmatrix" dialog (see figure below).
2. In the "Add External Switchmatrix" dialog:
   a) Specify the IP address or hostname of the switch matrix.
   b) Select "Identify" to auto-detect the matrix type ("Driver") of the remote switch matrix.
   If the firmware cannot identify the device (e.g. because it is temporarily unavailable), select the appropriate "Driver" manually.



Registering a switch matrix is only the first step. To make the external ports available to the vector network analyzer, the RF connections have to be configured, as described in Section 5.17.6.4, "Switch Matrix RF Connections dialog", on page 859.

Known Devices
Table with the discovered switch matrices.

"Scan Instruments" refreshes the table;  copies a discovered switch matrix to the table of "Configured Devices".

Remote command:
n/a

Configured Devices
Table displaying the registered switch matrices.

A distinction is made between those switch matrices that are used in RF connections (see Section 5.17.6.4, "Switch Matrix RF Connections dialog", on page 859) and those that are not.

- *Used* matrices are represented by inactive rows (grayed out).
- *Unused* matrices are represented by active table rows (colored).

For unused switch matrices that were manually registered via "Add Device", some properties can be changed by editing the corresponding table cells.

A matrix can always be unregistered using the  button of the respective row. This automatically deletes the related RF connections and renumbers the remaining test ports.

The following symbols (grayed out for *used* devices) indicate the status of the respective device:

- – The device is online (connected, switched on, ready to be used).
- – There is a problem with the device.
  This state can be caused by different problems:
  – General communication error
    In this case, check whether the device is properly connected to the configured interface.
  – Self test error
    In this case enable error logging for external devices (see "Log Errors" on page 845), and search the Error Log tab for self test error codes of the device.
- – The device driver does not match exactly the device type. Probably the device cannot be controlled properly. Measurements results are questionable.

For matrices with LAN interface, the button in the last column of the "Configured Devices" table opens the "Device LAN configuration" dialog which allows to read and modify their "IP Configuration" (see Section 5.17.6.3, "Device LAN Configuration dialog", on page 858).

The related switch matrix must be **online** and connected via USB (and not via Direct Control or LAN interface). Otherwise the button is disabled.

Remote command:
`SYSTem:COMMunicate:RDEVice:SMATrix:COUNt?`
`SYSTem:COMMunicate:RDEVice:SMATrix<Matr>:SERial?`

Scan Instruments
Performs an autodetection of "Known Devices" on the supported interfaces.

Note: Unintentional switchover to remote control

When using the NI-VISA library, ensure that the network analyzer itself is not listed as a network device in the Measurement & Automation Explorer. Otherwise, "Scan Instruments" sends an identification query (*IDN?), causing the analyzer to close the "External Switch Matrix" dialog (without executing "Scan Instruments") and to activate the remote screen.

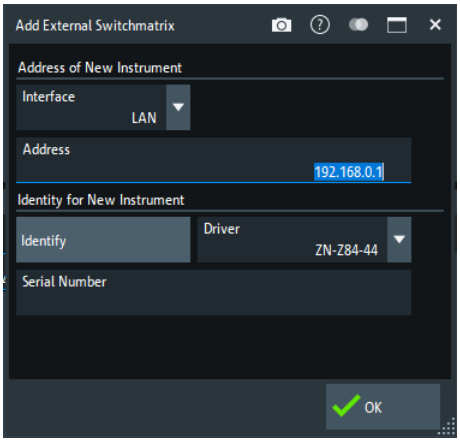
Remote command:
`SYSTem:COMMunicate:RDEVice:SMATrix:SCAN?`

LAN Detection
Activate "LAN Detection" to include the LAN interface in the autodetection sequence (see Scan Instruments).

Note: "LAN detection" only works for external devices sharing the same IP subnet with the R&S ZNB.
As a prerequisite, the R&S ZNB must accept incoming connections on UDP port 2473. If necessary, add a corresponding inbound rule to your firewall settings.

Remote command:
`SYSTem:COMMunicate:RDEVice:GDEVice:LAN:STATe`

Add Device
Opens the "Add External Switchmatrix" dialog that allows manual registration of an external switch matrix.



In the "Address of New Instrument" part, you can specify the required connection properties:

- "Interface" selects the connection type; currently only "LAN" is available.
- "Address" specifies the IP address or hostname of the external switch matrix.

If a connection can be established, the entries in the [Driver for New Instrument] panel can be auto-detected: "Identify" sends an identification query ("IDN?") to the specified device address to get the type and serial number of the connected switch matrix and to select the appropriate driver file.

Otherwise, manually select the "Driver". Matrix driver files (`*.matrix`) are stored in the `Resources\ExtDev` subdirectory of the analyzer's program directory.

Remote command:
`SYSTem:COMMunicate:RDEVice:SMATrix<Matr>:DEFine`
`SYSTem:COMMunicate:RDEVice:SMATrix<Matr>:SERial?`

Delete All
Unregisters all switch matrices, automatically deleting all switch matrix RF connections.

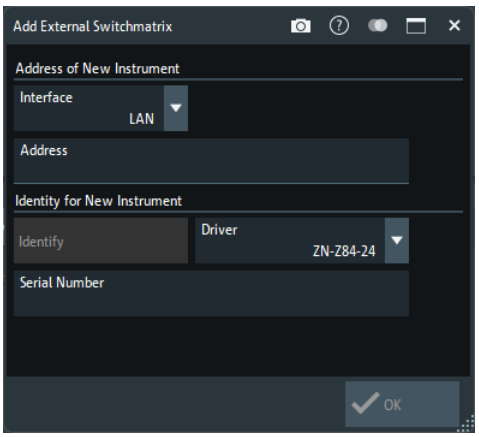
Remote command:
`SYSTem:COMMunicate:RDEVice:SMATrix:DELete`

5.17.6.3 Device LAN Configuration dialog

The "Device LAN Configuration" dialog allows you to read and to modify the "IP Configuration" of an external device that is equipped with a LAN interface.

Access: External Matrices dialog > "Configure"

Because the IP configuration is written to the device, the device must be **online** for the settings to be applied.



Disable "Manual Configuration" if you want the device to discover its host configuration via DHCP. In this case only the device's "Hostname" can be specified.

In "Manual Configuration" mode also the unit's "IP Address", "Subnet Mask" and "Standard Gateway" can be set. If the unit is located in the same subnet as the analyzer, simply use "Copy from local network" to copy the "Subnet Mask" and "Standard Gateway" from the analyzer's IP settings.

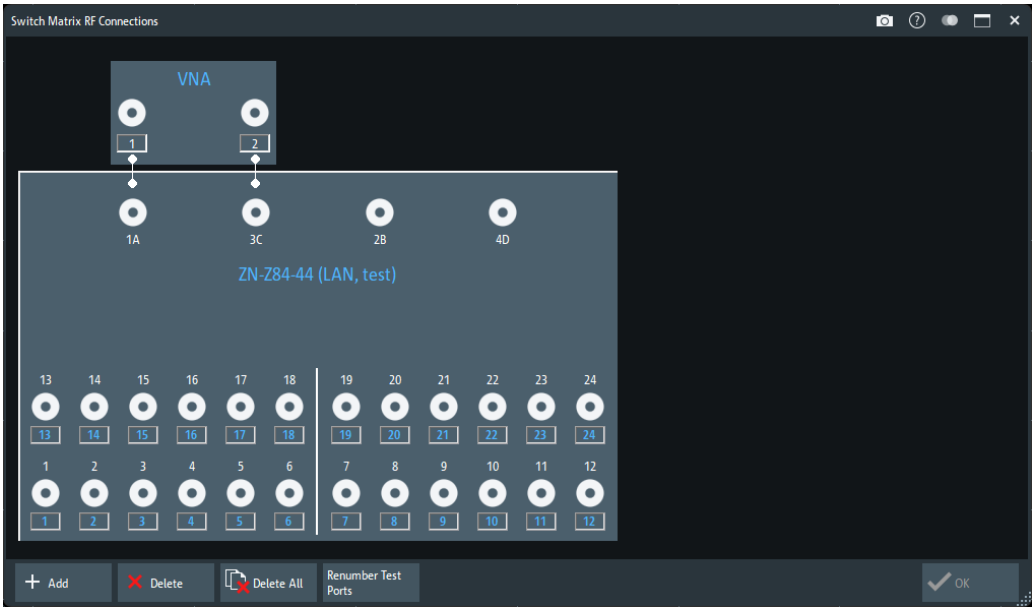
5.17.6.4 Switch Matrix RF Connections dialog

The "Switch Matrix RF Connections" dialog allows you to define the RF connections between the VNA and the switch matrices, and the test port numbering.

Access: System – [Setup] > "External Ports" > "RF Connections..."

Background information

Refer to section Section 4.7.30, "External switch matrices", on page 271.



Defining switch matrix RF connections typically involves the following steps.

1. Add one or more external switch matrices to the RF connection configuration. These matrices must have been registered before; see Section 5.17.6.2, "External Matrices dialog", on page 855 for details.
2. For each matrix VNA port (the "north" ports in the graphical representation), specify the connected VNA port or define it as "Unused" if no such connection exists.

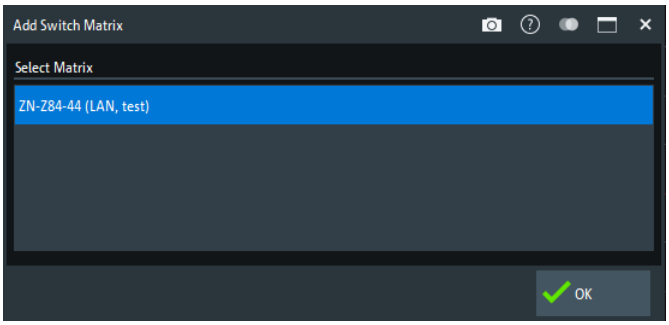
   For every matrix, there must be at least one RF connection to the VNA.
3. Select Renumber Test Ports to load the default test port assignment.

   Unusable matrix test ports are marked with an "x". If the result does not suit your needs, consider the switchable paths of your matrix and adjust the matrix VNA port connections (see Section 4.7.30.3, "RF connections and matrix connectivity", on page 273). Then select "Renumber Test Ports" again.

You can tweak the resulting test port assignment by renumbering the test ports and/or setting some of the matrix test ports to "Unused" (see "Edit Test Port Connection" on page 862). Configuration errors are displayed in balloon popups appearing at the lower left corner of the dialog.

A redefinition of the physical VNA ports (see Section 5.17.8.2, "Define Physical Ports dialog", on page 864) causes a factory reset and deletes all switch matrix RF connections. So the RF configuration for switch matrices has to be done *after* the port redefinition.

Add
Opens the "Add Switch Matrix" dialog:

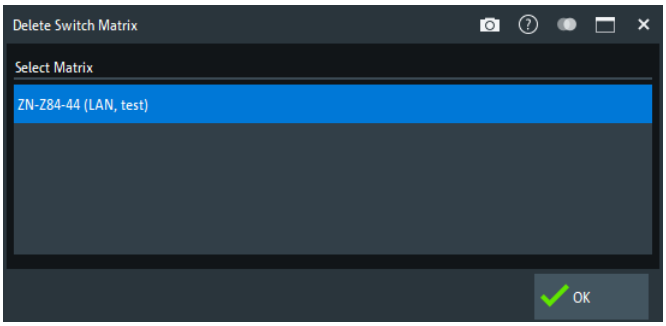


Select the required matrices and tap "OK" to add them to the RF connection configuration or tap "Cancel" to quit the dialog without adding a switch matrix to the RF connection configuration

Note:
- This dialog is shown automatically, in case no matrix has been added to the RF connection configuration yet.
- If the RF configuration already contains all registered switch matrices, the "Add" button is disabled.

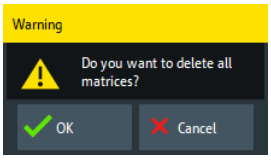
Remote command:
n.a.

Delete
Opens the "Delete Switch Matrix" dialog:



Select some matrices and tap "OK" to remove them from the RF connection configuration or tap "Cancel" to quit the dialog without removing a switch matrix from the RF connection configuration.

Delete All
Opens a confirmation dialog



Either select "OK" to remove all switch matrices from the RF connection configuration, or "Cancel" to quit the dialog without removing a switch matrix.

Remote command:
`SYSTem:COMMunicate:RDEVice:SMATrix:DELete`

Renumber Test Ports
Use this function at any time to generate the default test port assignment for the given matrix VNA port connections.

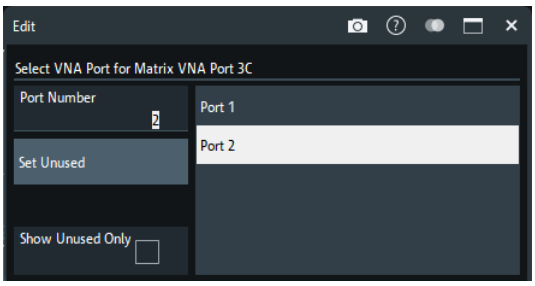
The following RF ports are used as test ports (and numbered from 1 to N):

- All matrix test ports that can be switched to at least one of the connected matrix VNA ports
- All VNA ports that are not connected to a switch matrix

Note that "Renumber Test Ports" does not preserve manually assigned test port numbers or "Unused" flags. See "Edit Test Port Connection" on page 862.

Remote command:
n.a.

Edit Matrix VNA Port Connections
To edit a matrix VNA port connection, tap on the corresponding port symbol.



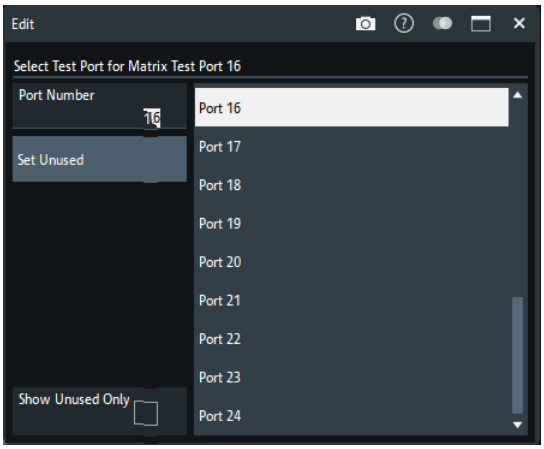
In the "Edit" dialog, define the appropriate VNA port or "Set Unused" to delete the matrix VNA port connection.

Remote command:

```
SYSTem:COMMunicate:RDEVice:SMATrix<Matr>:CONFigure:MVNA
SYSTem:COMMunicate:RDEVice:SMATrix<Matr>:CONFigure:MLVNa
```

Edit Test Port Connection

To edit a test port connection, tap on the corresponding port symbol.



In the "Edit" dialog, define the appropriate test port or select "Set Unused" to delete the test port assignment. Configuration errors are displayed in balloon popups appearing at the lower left corner of the "Switch Matrix RF Connections" dialog.

Remote command:

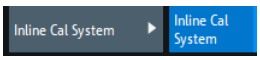
```
SYSTem:COMMunicate:RDEVice:SMATrix<Matr>:CONFigure:MTESt
SYSTem:COMMunicate:RDEVice:SMATrix<Matr>:CONFigure:MLTest
SYSTem:COMMunicate:RDEVice:SMATrix:CONFigure:TVNA
```

5.17.7 Inline Cal System tab

The "Inline Cal System" tab gives access to inline calibration systems connected to the R&S ZNB via USB.

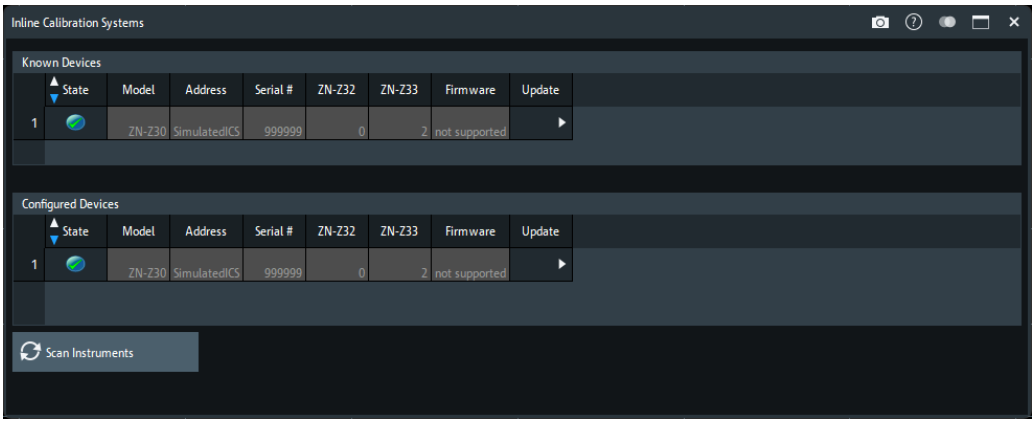
Background Information

See Section 4.5.5.4, "Inline calibration", on page 179



5.17.7.1 Inline Calibration Systems dialog

USB-connected Inline Calibration Controllers (ICCs) are autodetected by the analyzer firmware.



- The ICC does not support hot plugging of ICUs, so connect the ICUs to the ICC before connecting the ICC to the R&S ZNB.
- For background information, see Section 4.5.5.4, "Inline calibration", on page 179.

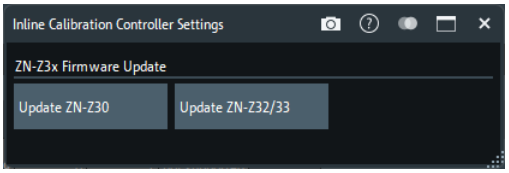
Known Devices/Configured Devices
Each ICC R&S ZN-Z30 is listed with its connected Inline Calibration Units (ICUs) R&S ZN-Z32 and R&S ZN-Z33.

ZN-Z32/ZN-Z33 ← Known Devices/Configured Devices
Each ICC R&S ZN-Z30 is listed with its connected Inline Calibration Units (ICUs) R&S ZN-Z32 and R&S ZN-Z33.

Remote command:
`SYSTem:COMMunicate:RDEVice:AKAL:ADDRess:SUBModule:ALL?`

Firmware/Update
"Firmware" displays the firmware installed on the ICC.

The ellipsis button in the "Update" column opens a dialog that allows you to update the firmware of the respective ICC (R&S ZN-Z30) and ICUs (R&S ZN-Z32/33).

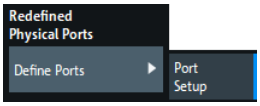


The firmware of the R&S ZNB includes the latest firmware binaries available for the ICC and the ICUs. They are available at `C:\Program Files\Rohde-Schwarz\Vector Network Analyzer\ZNB\Resources\ICS`. Select the related "Update ZN-Z3..." button to browse this directory for the suitable `*.bin` file and to start the update process.

5.17.8 Port Setup tab

Collects functions for the (re) definition of internal and external physical ports and their signal paths.

5.17.8.1 Controls on the Port Setup tab



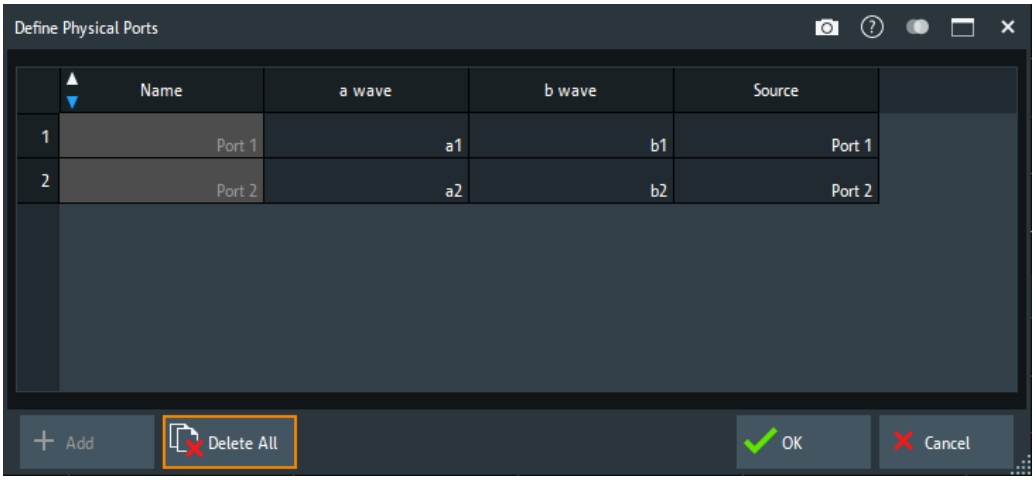
Define Ports...
Opens a dialog that allows you to redefine the physical VNA ports. See Section 5.17.8.2, "Define Physical Ports dialog", on page 864.

5.17.8.2 Define Physical Ports dialog

Allows you to redefine the physical VNA ports (see Section 4.3.1.2, "Redefined S-parameters", on page 118).

Access: System – [Setup] > "External Ports" > "Define Ports..."

- Redefined physical ports are global, persistent settings, i.e. they are valid for all recall sets. A [Preset] does not reset the physical port configuration.
- Redefining physical ports causes a factory reset.
  The factory reset deletes all switch matrix RF connections.



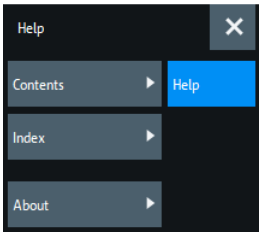
a wave, b wave, Source
Define a physical port by assigning its reference receiver, measurement receiver and generator, respectively.

The receivers and generators can be freely assigned, but without reusing the same (original) physical port in different (redefined) ports.

Remote command:
`[SENSe:]UDSParams<Pt>:PARam`
`[SENSe:]UDSParams:ACTive`

5.18 Help softtool

The "Help" softtool presents the functions of the menu bar's "Help" menu, except calling context sensitive help.



Access: "Help" – "Contents..." | "Index..." | "About..." from the menu bar.

Contents...
Opens the help browser and activates its "Contents" tab.

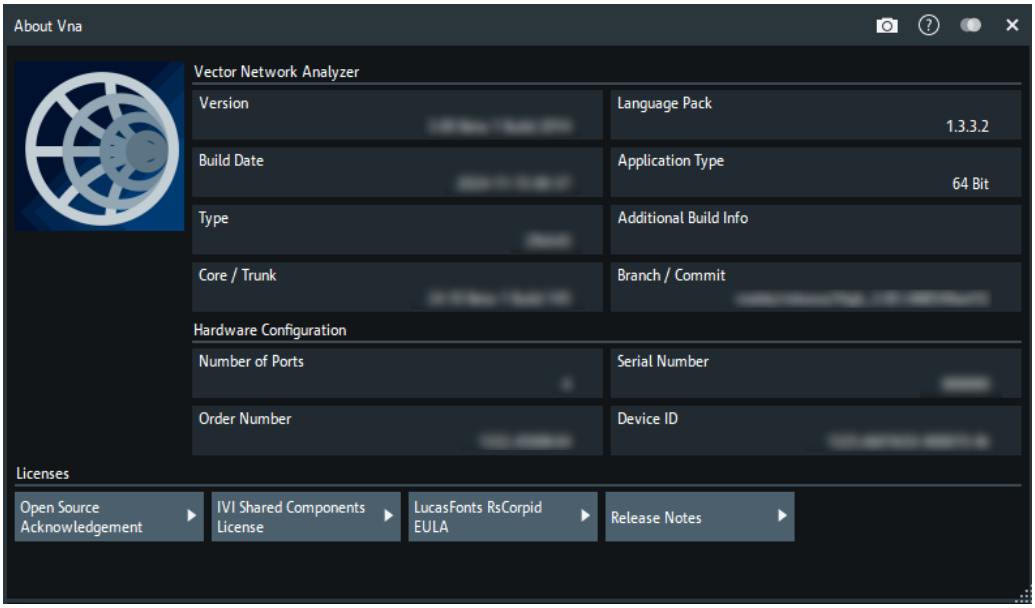
The "Contents" tab displays the table of contents of the analyzer's online help.

Index...
Opens the help browser and activates its "Index" tab.

The "Index" tab displays the index keywords of the analyzer's online help.

About...
Opens the "About Vna" dialog.

The "About Vna" provides basic information about the analyzer model and the current firmware. It also gives access to related license agreements.



Open the Info dialog for full information about the instrument.

5.19 Additional function keys

The System hardkey panel also provides the Windows®, [Help] and [Preset] keys that do not open a softtool but perform an immediate action.

The data entry panel provides hardkeys for undo and redo.

System – [Preset]
Performs a preset of all instrument settings or of the active recall set, depending on the settings in the Presets tab of the "System Config" dialog.

A preset can be a factory preset or a user-defined preset. It does not affect global settings and resources (e.g. "System Config" settings, calibration kits and calibration pool etc.).

Tip: If you perform a [Preset] by mistake, use [Undo] to restore your previous instrument settings.

Remote command:

```
*RST
SYSTem:PRESet:SCOPe
SYSTem:PRESet:USER:NAME
SYSTem:PRESet:USER[:STATe]
SYSTem:PRESet[:DUMMy]
```

data entry – [Undo] / [Redo]
[Undo] reverses the last action, [Redo] reverses a preceding [Undo].

In the Toolbar, the corresponding icons are disabled (grayed out) if an undo or redo is not possible. In general, the undo and redo actions are disabled if the size of the active recall set file exceeds 1 MB.

Tip: You can also use [Undo] after a System – [Preset], to restore your previous instrument settings.

6 Remote control

This chapter provides instructions on how to set up the analyzer for remote control, a general introduction to remote control of programmable instruments, and the description of the analyzer's remote control concept. For reference information about all remote control commands implemented by the instrument, complemented by comprehensive program examples, refer to Section 7, "Command reference", on page 911.

For additional information on remote control of network analyzers, see the following documents available from the Rohde & Schwarz website:

- Remote control via SCPI
- 1EF62: Hints and Tricks for Remote Control of Spectrum and Network Analyzers
- 1MA171: How to use Rohde & Schwarz Instruments in MATLAB
- 1MA208: Fast Remote Instrument Control with HiSLIP

6.1 Introduction to remote control

The instrument offers different interfaces for remote control:

- **LAN**
  Analyzers connected to a LAN can be remote-controlled via the RSIB, VXI-11, or HiSLIP protocols. The LAN connector is on the rear panel of the instrument see (see Section 3.2.2, "Rear panel", on page 41). A VISA installation on the remote control PC is required.
- **USB**
  The R&S ZNB can also be remote-controlled via USB. The USB Device connector is on the rear panel of the instrument. A VISA installation on the remote control PC is required.
- **GPIB**
  With option R&S ZNB-B10, the analyzer offers a GPIB bus interface according to standard IEC 625.1/IEEE 488.1. If equipped, the GPIB bus connector is on the rear panel of the instrument. The R&S ZNB can be remote-controlled via GPIB.
- The network analyzer can itself act as a master and control external devices via LAN, USB, or GPIB. A VISA installation on the analyzer is a prerequisite for this type of remote control. The RS VISA library is installed with the VNA firmware.
  - To control external devices via USB, the "IVI Visa Shared Components" must be installed in addition. You can easily install it from the "Start" menu of your analyzer.
  - To control external devices via GPIB, you have to use an Section 4.7.27, "USB-to-IEC/IEEE adapter", on page 267.

VISA library

VISA is a standardized software interface library providing input and output functions to communicate with instruments. The I/O channel (LAN, USB, GPIB, ...) is selected at initialization time by its channel-specific resource string (also termed address string), or by an appropriately defined VISA alias (short name).

For more information about VISA, refer to the installed library's user documentation.

HiSLIP protocol

The HiSLIP (High Speed LAN Instrument Protocol) is a protocol for TCP-based instruments specified by the IVI foundation. Compared to its predecessor VXI-11, it provides speed and other improvements. HiSLIP is encapsulated in VISA; the resource string reads `TCPIP::<R&S ZNB IP address>::hislip0`.

The RS Visa library supports HiSLIP. If the connection fails, access the Windows control panel of the controlled instrument and open port 4880 for incoming connections.

6.1.1 Remote control via USB

The instrument can be remote-controlled via USB. The VISA resource string for controlling a device via USB has to be specified in the format

`USB[board]::<Manufacturer ID>::<Device ID>::<Serial Number>::INSTR`.

The `<Manufacturer ID>` of Rohde & Schwarz is `0x0AAD`, the `<Device ID>` is model-specific.

Table 6-1: R&S ZNB Device IDs

	1st generation (1311.6010Kxx)	2nd generation (1334.3330Kxx)
2-port R&S ZNB4	0x019C	0x01E4
4-port R&S ZNB4	0x019D	0x01E5
2-port R&S ZNB8	0x019E	0x01E6
4-port R&S ZNB8	0x019F	0x01E7
2-port R&S ZNB20	0x01A0	0x01E8
4-port R&S ZNB20	0x01A1	0x01E9
2-port R&S ZNB26	-	0x0297
4-port R&S ZNB26	-	0x0298
2-port R&S ZNB40 var. 72	0x01A2	-
2-port R&S ZNB40 var. 82	0x01BF	-
2-port R&S ZNB40 var. 83	0x0263	-
4-port R&S ZNB40 var. 84	0x01C0	-
2-port R&S ZNB43 2.92 mm	-	0x025B

	1st generation (1311.6010Kxx)	2nd generation (1334.3330Kxx)
4-port R&S ZNB43 2.92 mm	-	0x025C
2-port R&S ZNB43 2.4 mm	-	0x0268
4-port R&S ZNB43 2.4 mm	-	0x0269
R&S ZNBT8	0x01BE	-
R&S ZNBT20	0x01A7	-
R&S ZNBT26	0x01F3	-
R&S ZNBT40	0x01F4	-

Hence a possible resource string for the R&S ZNBT20 is `USB1::0x0AAD::0x01A7::100067::INSTR`.

6.1.2 Starting a remote control session

A remote control program must open a connection to the analyzer (using VISA functionality), before it can send commands to the analyzer and receive device responses (e.g. measurement data). The programming details depend on the library version used and on the programming language. For this reason, the examples in chapters "Command Reference" and "Programming Examples" are reduced to the mere SCPI syntax.

Example controller programs can be obtained from the Rohde & Schwarz support centers. However, it can be preferable to integrate the controller program into post-processing tools (e.g. Microsoft Excel) to list, draw, or manipulate the measured values retrieved from the analyzer.

The following tools can make remote control more comfortable and faster:

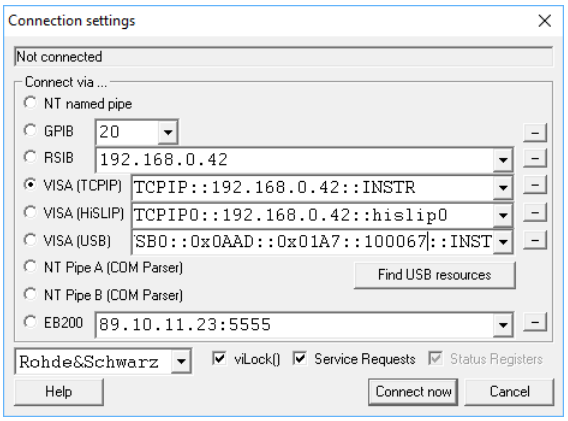
- Various software tools provide an easy-to-use graphical user interface for remote control. An example is the "GPIB Explorer" (also termed "IECWIN32") which is preinstalled on the analyzer. See Section 6.1.3, "GPIB Explorer", on page 870.
- Instrument drivers provide an improved interface between the test software and the test instruments. They perform the actual control of the instrument using higher-level functions for operations such as configuring, reading from, writing to, and triggering the instrument. Hence, drivers can reduce development time, eliminating the need to learn the specific command set for each instrument. In general, program development is further simplified by a graphical program environment.
  Rohde & Schwarz offers various R&S ZNB driver types (LabView, LabWindows/CVI, IVI, VXIplug&play...) for different programming languages. The drivers are available free of charge on the product pages in the R&S internet, along with installation information.

6.1.3 GPIB Explorer

The GPIB Explorer is a software tool that allows you to connect to the analyzer, and to obtain an overview of all implemented remote control commands. You can use it to

compile and run test scripts. The program can be opened from the Windows® start menu: "Programs – R&S ZNB Vector Network Analyzer – GPIB Explorer" or via System – [Applic] "External Tools" > "GPIB Explorer". You can also start the executable file `iecwin32.exe` in the program directory of the network analyzer (e.g. `C:\Program Files\Rohde-Schwarz\Vector Network Analyzer\ZNB\Bin`).

After the GPIB Explorer is started, the interface and protocol for the connection to the instrument can be selected in a dialog:

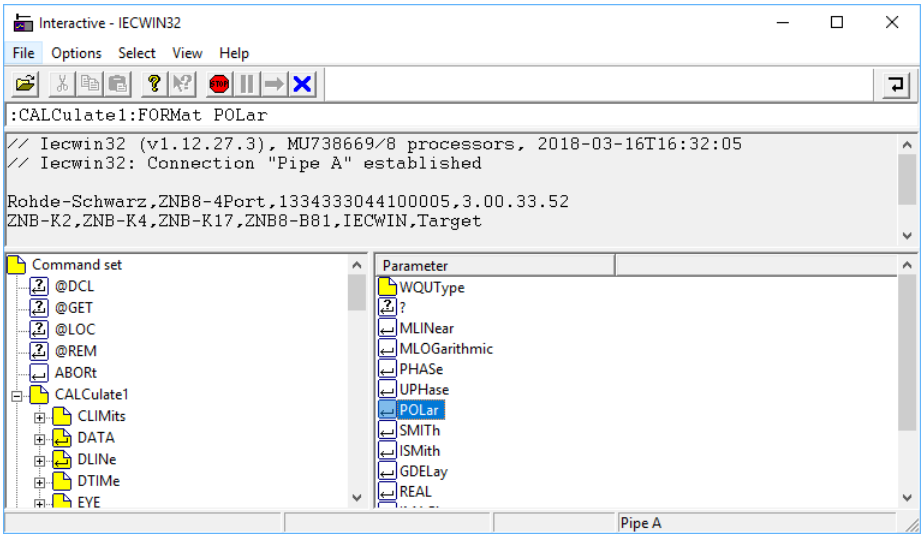


The following options are provided:

- **NT named pipe** (currently not supported)
- **GPIB address** (for connection to controllers equipped with a National Instruments GPIB interface using the GPIB bus connector)
- **RSIB**, **VISA (TCPIP)** and **VISA (HSLIP)** (for LAN connection, requires an appropriate IP or local host address); see Section 3.1.12.1, "Assigning an IP address", on page 31.
- **VISA (USB)**
  Depending on the instrument model and variant, specify the resource string as `USB1::0x0AAD::<Device ID>::<Serial>::INSTR` (see Section 6.1.1, "Remote control via USB", on page 869).
- **NT pipe A/B (COM Parser)** (only for a GPIB Explorer installed on the analyzer, recommended for "remote" test on the instrument)
- **EB200** (currently not supported)

Select "SETUP > Setup > Info..." to look up the IP address information of your analyzer. If you run the GPIB explorer on the analyzer, the local host address (loopback address) is 127.0.0.1.

After the connection is established, the GPIB explorer displays a tree view of all commands included in the current firmware version of the network analyzer. The programs can be selected for execution by a single mouse click.

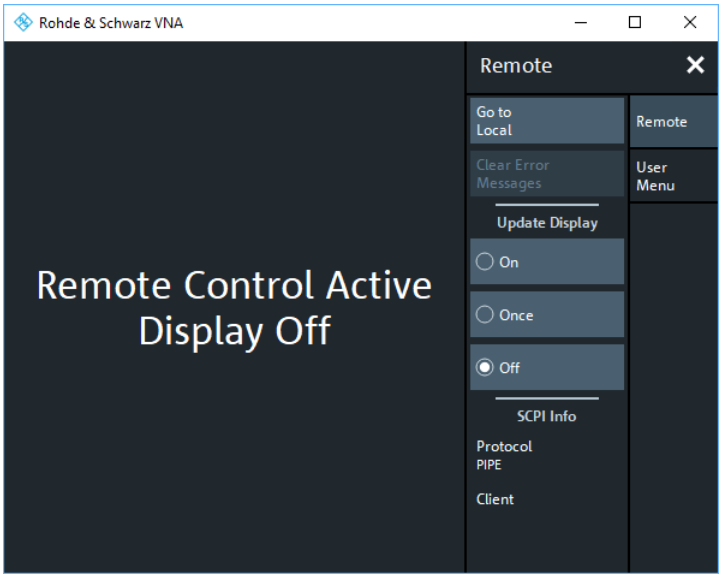


It remote logging is enabled (`SYSTem:LOGGing:REMote[:STATe] ON`) the analyzer stores all received commands to the file `'C:\Users\Public\Documents\Rohde-Schwarz\Vna\RemoteLog'`.

6.1.4 Switchover to remote control

On power-up, the instrument is always in the manual operating state and can be operated via the front panel controls. The instrument is switched to remote control when it receives a command from the controller. If the instrument is controlled via RSIB or VXI-11 protocol, the alternative commands @REM and @LOC can be used to switch from manual to remote control and back.

While remote control is active, operation via the front panel is disabled except the "Remote" softtool. The instrument settings are optimized for maximum measurement speed; the display is switched off:



In case the "Remote" softtool is not visible, tap/click on the toucscreen/application window to restore it.

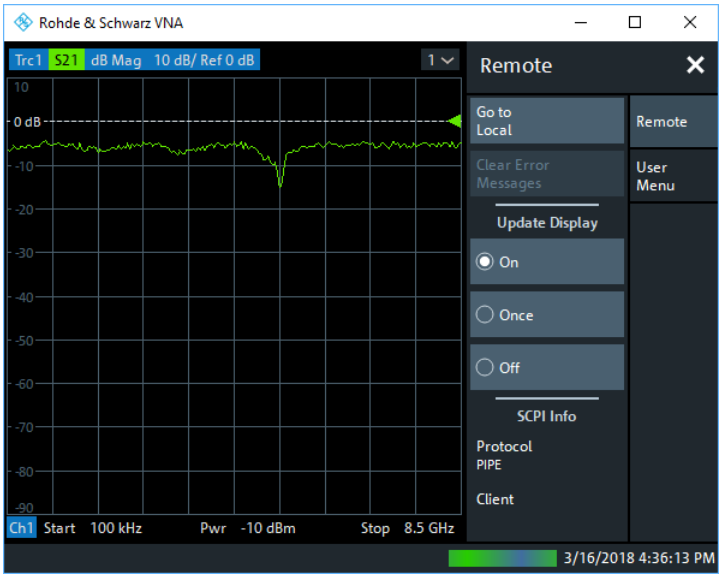
The softkeys in the remote screen are used to modify or quit the remote state:

- "Go to Local" switches the instrument to local state.
- "Display" switches the display on or off.
- If a remote error message is displayed at the bottom of the remote screen, you can use "Clear Error Messages" to delete it.

The remaining controls are for future extensions.

Display on and off states

Switching on the display means that the analyzer shows the measurement screen with the current recall sets, diagram areas and traces without leaving the remote state. In this operating mode, it is possible to observe the screen while a remote control script is executed and the control elements on the front panel are still disabled.

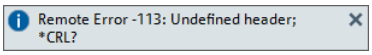


Switching on the display is ideal for program test purposes but tends to slow down the measurement. Therefore, it is recommended to switch off the display in real measurement applications where a tested program script is to be executed repeatedly.

The analyzer provides a third display option where the measurement screen is only updated when triggered by the remote control command `SYSTem:DISPlay:UPDate ONCE`.

The instrument remains in the remote state until it is reset to the manual state via the GUI or via remote control (see Section 6.1.4.2, "Returning to manual operation", on page 875). You can also lock the remote (touch) screen using `SYSTem:TSLock SCReen`.

A tooltip across the bottom of the remote screen indicates a remote command error. You can switch off this tooltip using `SYSTem:ERRor:DISPlay:REMote OFF`.



SCPI commands:

`@REM`

`SYSTem:DISPlay:UPDate`

`SYSTem:TSLock`

`SYSTem:ERRor:DISPlay[:REMote]`

6.1.4.1 Setting the device address

The GPIB address (primary address) of the instrument is factory-set to 20. It can be changed manually in the System – [Setup] > "Remote Settings" tab or via remote con-

trol. For remote control, addresses 0 through 30 are permissible. The GPIB address is maintained after a reset of the instrument settings.

SCPI commands:

`SYSTem:COMMunicate:GPIB[:SELF]:ADDRess`

6.1.4.2 Returning to manual operation

Return to manual operation can be initiated via the front panel or via remote control.

- Manually: tap the Local softkey in the remote screen.
- Via GPIB bus: CALL IBLOC(device%)
- Via RSIB or VXI-11 protocol: @LOC and @REM can be used to switch from remote to manual control and back.



Local lockout

Before returning to manual control, command processing must be completed. If not, the analyzer switches back to remote control immediately.

Returning to manual control by tapping the "Go to Local" softkey can be disabled, e.g., by the Local Lockout Message (via GPIB: LLO; see Section 13.3.3.2, "Interface messages", on page 1706). The lockout prevents unintentional switch-over, i.e. return to manual control is possible via remote control only.

Returning to manual control via the front panel keys can be enabled again, e.g. by deactivating the REN control line of the GPIB bus.

6.1.5 Combining manual and remote control

Using a remote control script is the quickest and easiest way of performing complicated tasks which need to be repeated many times. However, it is often preferable to control a previously configured measurement manually to observe the result on the screen.

The analyzer provides various tools for combining manual and remote control:

- **User Keys**
  The remote control commands `SYSTem:USER:KEY...` place up to 8 softkeys with arbitrary functionality on the remote screen. The softkeys appear in the "User Menu" tab of the "Remote" softtool. When a softkey is selected, the ESR bit no. 6 (User Request) is set, and the response for `SYSTem:USER:KEY?` is changed. This behavior can serve as a control mechanism in remote control scripts.

SCPI commands:

`SYSTem:USER:KEY`

6.2 Messages

The messages transferred on the data lines of the GPIB bus or via the RSIB / VXI-11 protocol can be either interface messages or device messages. For a description of interface messages refer to the relevant sections:

- Section 13.3.3, "GPIB interface", on page 1704
- Section 13.3.2, "LAN interface", on page 1703

6.2.1 Device messages (commands and device responses)

Depending on the selected "Codec" (see `SYSTem:COMMunicate:CODec` on page 1539), device messages are either transferred in ANSI, UTF-8 or Shift JIS format. A distinction is made according to the direction in which device messages are transferred:

- Commands are messages that the controller sends to the instrument. They operate the device functions and request information.
- Device responses are messages that the instrument sends to the controller after a query. They can contain measurement results, instrument settings and information on the instrument status.

Commands are subdivided according to two criteria:

1. According to the effect they have on the instrument:
   - Setting commands cause instrument settings such as a reset of the instrument or setting the output level to some value.
   - Queries cause data to be provided for output on the GPIB bus, e.g. for identification of the device or polling the active input.
2. According to their definition in standard IEEE 488.2:
   - Common commands have a function and syntax that is precisely defined in standard IEEE 488.2. Typical tasks are the management of the standardized status registers, reset and selftest.
   - Instrument-control commands are functions that depend on the features of the instrument such as frequency settings. Most of these commands have also been standardized by the SCPI consortium.

The device messages have a characteristic structure and syntax. In the SCPI reference chapter, all commands are listed and explained in detail.

6.2.2 SCPI command structure and syntax

SCPI commands consist of a so-called header and, usually, one or more parameters. The header and the parameters are separated by a white space (ASCII code 0 to 9, 11 to 32 decimal, e.g. blank). The headers can consist of several mnemonics. Queries are formed by directly appending a question mark to the header.

Common commands and device-specific commands differ in their syntax.

SCPI compatibility

The analyzers are compatible to the final SCPI version 1999.0. Not all the commands supported by the instrument are taken from the SCPI standard (Standard Commands for Programmable Instruments), however, their syntax follows SCPI rules. The SCPI standard is based on standard IEEE 488.2 and aims at the standardization of instrument-control commands, error handling and the status registers.

The requirements that the SCPI standard places on command syntax, error handling and configuration of the status registers are explained in detail in the following sections.

Reset values

In contrast to instruments with manual control, which are designed for maximum operating convenience, the priority of remote control is the predictability of the device status. This means that when incompatible settings are attempted, the command is ignored and the device status remains unchanged, i.e. other settings are not automatically adapted. Therefore, GPIB bus control programs should always define an initial device status (e.g. with the command *RST) and then implement the required settings.

6.2.2.1 Common commands

Common (=device-independent) commands consist of a header preceded by an asterisk "*" and possibly one or more parameters.

Examples:	
*RST	RESET resets the instrument.
*ESE 253	EVENT STATUS ENABLE sets the bits of the event status enable registers.
*ESR?	EVENT STATUS QUERY queries the contents of the event status register.

6.2.2.2 Instrument-control commands

Instrument-control commands are based on a hierarchical structure and can be represented in a command tree. The command headers are built with one or several mnemonics (keywords). The first level (root level) mnemonic identifies a complete command system.

Example:

`SENSe` This mnemonic identifies the command system `SENSe`.

For commands of lower levels, the complete path has to be specified, starting on the left with the highest level, the individual mnemonics being separated by a colon ":".

Example:

```
SENSe:FREQuency:STARt 1GHZ
```

This command is on the third level of the `SENSe` system. It defines the start frequency of the sweep.

The following rules simplify and abbreviate the command syntax:

- **Multiple mnemonics**
  Some mnemonics occur on several levels within one command system. Their effect depends on the structure of the command, i. e. on the position in the command header they are inserted in.
  Example:
  `SOURce:FREQuency:CW 1GHZ`
  This command contains the mnemonic `SOURce` in the first command level. It defines the frequency for sweep types operating at fixed frequency.
  `TRIGger:SOURce EXTernal`
  This command contains the mnemonic `SOURce` in the second command level. It defines the trigger source “external trigger”.
- **Optional mnemonics**
  Some command systems permit certain mnemonics to be optionally inserted into the header or omitted. These mnemonics are marked by square brackets in this manual. The full command length must be recognized by the instrument for reasons of compatibility with the SCPI standard. Some commands are considerably shortened by omitting optional mnemonics.
  Example:
  `TRIGger[:SEQuence]:SOURce EXTernal`
  This command defines the trigger source “external trigger”. The following command has the same effect:
  `TRIGger:SOURce EXTernal`
  Note:
  The short form is marked by uppercase letters, the long form corresponds to the complete word. Uppercase and lowercase notation only serves to distinguish the two forms in the manual, the instrument itself is case-insensitive.
- **Parameters**
  Parameters must be separated from the header by a white space. If several parameters are specified in a command, they are separated by a comma ",". For a description of the parameter types, refer to section Parameters.
  Example:
  `SOURce:GROup 1,1`
  This command defines a group of measured ports.
- **Numeric suffix**
  If a device features several functions or features of the same kind, e.g. several channels or test ports, the desired function can be selected by a suffix added to the command. Entries without suffix are interpreted like entries with the suffix 1.
  Example:
  `SOURce:GROup2 1,1`
  This command defines a second group (group no 2) of measured ports.

6.2.2.3 Structure of a command line

A command line can consist of one or several commands. It is terminated by a <New Line>, a <New Line> with EOI or an EOI together with the last data byte. Tools like the GPIB Explorer automatically produce an EOI together with the last data byte.

Several commands in a command line must be separated by a semicolon ";". If the next command belongs to a different command system, the semicolon is followed by a colon.

Example: `TRIGger:SOURce EXTernal;:SENSe:FREQuency:STARt 1GHZ`

This command line contains two commands. The first command belongs to the `TRIGger` system and defines the trigger source (external trigger). The second command belongs to the `SENSe` system and defines the start frequency of the sweep.

If the successive commands belong to the same system, having one or several levels in common, the command line can be abbreviated. To this end, the second command after the semicolon starts with the level that lies below the common levels. The colon following the semicolon must be omitted in this case.

Example: `TRIG:SOUR EXT;:TRIG:TIM 0.1`

This command line is represented in its full length and contains two commands separated from each other by the semicolon. Both commands are part of the `TRIGger` command system, i.e. they have one level in common.

When abbreviating the command line, the second command begins with the level below `TRIG`. The colon after the semicolon is omitted. The abbreviated form of the command line reads as follows:

```
TRIG:SOUR EXT; TIM 0.1
```

However, a new command line always begins with the complete path.

Example:	`TRIG:SOUR EXT` `TRIG:THR LOW`

6.2.2.4 Responses to queries

A query is defined for each setting command unless explicitly specified otherwise. It is formed by adding a question mark to the associated setting command. According to SCPI, the responses to queries are partly subject to stricter rules than in standard IEEE 488.2.

1. The requested parameter is transmitted without header.
   Example: `TRIGger:SOURce?` Response: `IMM`
2. Maximum values, minimum values and all further quantities which are requested via a special text parameter are returned as numerical values.
   Example: `SENSe:FREQuency:STOP? MAX` Response: `8000000000`
3. Numerical values are output without their unit. The default unit for each command is reported in the SCPI command description.
   Example: `SENSe:FREQuency:STOP? MAX` Response: `8000000000` for 8 GHz
4. Boolean values are returned as 0 (for OFF) and 1 (for ON).
   Example: `SWEep:TIME:AUTO?` Response: `1`
5. Text (character data) is returned in short form (see also next section).

Example: TRIGger:SOURce? Response: IMM

6.2.3 SCPI parameters

Many commands are supplemented by a parameter or a list of parameters. The parameters must be separated from the header by a "white space". Permissible parameters are numerical values, Boolean parameters, text, character strings and block data. The type of parameter required for the respective command and the permissible range of values are specified in the command description.

6.2.3.1 Numeric values

Numeric values can be entered in any form, i.e. with sign, decimal point and exponent. Values exceeding the resolution of the instrument are rounded up or down. The mantissa can comprise up to 255 characters, the values must be in the value range –9.9E37 to 9.9E37. The exponent is introduced by an E or e. Entry of the exponent alone is not allowed. For physical quantities, the unit can be entered. Permissible unit prefixes are G (giga), MA (mega), MOHM and MHZ are also permissible), K (kilo), M (milli), U (micro) and N (nano). If the unit is missing, the default unit is used.

Example:

SOUR:RFG:FREQ 1.5GHz is equivalent to:

SOUR:RFG:FREQ 1.5E9

Special numeric values

The texts MINimum, MAXimum, DEFault, UP and DOWN are interpreted as special numeric values. A query returns the associated numerical value.

Example:

Setting command: SENSe:FREQuency:STARt MINimum

The query SENSe:FREQuency:STARt? returns 300000 (the exact value depends on the analyzer model).

The following special values can be used:

- MIN/MAX MINimum and MAXimum denote the minimum and maximum value of a range of numeric values.
- DEF DEFault denotes the preset value. This value is set by the *RST command.
- **UP/DOWN** UP, DOWN increases or reduces the numeric value by one step. The step width is reported in the detailed command description.
- INF/NINF Negative INFinity (NINF) represent the numerical values –9.9E37 or +9.9E37, respectively. INF and NINF are only sent as device responses.
- NAN Not a Number (NAN) represents the value 9.91E37. NAN is only sent as device response. This value is not defined. Possible causes are division by zero, subtraction or addition of infinite and the representation of missing values.

Unless it is explicitly stated in the command description, you can use the special numeric parameters for all commands of the analyzer.

6.2.3.2 Boolean parameters

Boolean parameters represent two states. The **ON** state (logically true) is represented by **ON** or a numerical value different from 0. The **OFF** state (logically false) is represented by **OFF** or the numerical value 0. A query responds with 0 or 1.

`Example:` Setting command: `SWEep:TIME:AUTO ON`

Query: `SWEep:TIME:AUTO?` returns 1

6.2.3.3 Text parameters

Text parameters observe the syntax rules for mnemonics, i.e. they can be entered using a short or long form. Like any parameter, they have to be separated from the header by a white space. Queries return the short form of the text.

Example: Setting command: `TRIGger:SOURce EXTernal`

Query: `TRIGger:SOURce?` returns `EXT`

6.2.3.4 Strings

Strings must always be entered within single or double quotation marks (' or ").

Example: `CONFigure:CHANnel:NAME "Channel 4"` or

`CONFigure:CHANnel:NAME 'Channel 4'`

6.2.3.5 Block data format

Block data is a transmission format which is suitable for the transmission of large amounts of data. A command using a block data parameter with definite length has the following structure:

Example: `HEADer:HEADer #45168xxxxxxxx`

The hash symbol # introduces the data block. The next number indicates how many of the following digits describe the length of the data block. In the example the 4 following digits indicate the length to be 5168 bytes. The data bytes follow. During the transmission of these data bytes all End or other control signs are ignored until all bytes are transmitted.

A #0 combination introduces a data block of indefinite length. The use of the indefinite format requires a NL^END message to terminate the data block. This format is useful when the length of the transmission is not known or if speed or other considerations prevent segmentation of the data into blocks of definite length.

6.2.3.6 Overview of syntax elements

:	The colon separates the mnemonics of a command. In a command line, the separating semicolon marks the uppermost command level.
;	The semicolon separates two commands of a command line. It does not alter the path.
,	The comma separates several parameters of a command.
?	The question mark forms a query.
*	The asterisk marks a common command.
', "	Quotation marks introduce a string and terminate it.
#	The hash sign # introduces binary, octal, hexadecimal and block data. Binary: #B10110 Octal: #O7612 Hexadecimal: #HF3A7 Block: #21312
	A "white space" (ASCII-Code 0 to 9, 11 to 32 decimal, e.g. blank) separates header and parameter.

6.3 Basic remote control concepts

The functionality of the network analyzer's remote control commands has been defined in close analogy to the menu commands and control elements of the graphical user interface (GUI). The basic concepts of recall sets, traces, channels, and diagram areas remain valid in remote control. Moreover, all commands follow SCPI syntax rules, and SCPI-confirmed commands have been used whenever possible. These principles largely simplify the development of remote control scripts.

The GUI and the remote control command set both aim at maximum operating convenience. For manual control, this generally means that the control elements are easy to find and intuitive to handle, and that the effect of each operation is easy to verify on the screen. Convenient remote control operation depends on a simple and systematic program syntax and on a predictable instrument state; the display of results is secondary.

These differences suggest the peculiarities in the analyzer's remote control concept discussed in the following sections.

6.3.1 Traces, channels, and diagram areas

Like in manual control, traces can be assigned to a channel and displayed in diagram areas (see section Traces, Channels and Diagram Areas in Chapter 3). There are two main differences between manual and remote control:

- A trace can be created without being displayed on the screen.
- A channel must not necessarily contain a trace. Channel and trace configurations are independent of each other.

The following frequently used commands create and delete traces, channels, and diagram areas:

Create new trace and new channel (if channel <Ch> does not exist yet)	CALCulate<Ch>:PARameter:SDEFine '<Trace Name>', '< Meas Parameter>'
Delete trace	CALCulate<Ch>:PARameter:DELete '<Trace Name>'
Create or delete channel	CONFigure:CHANnel<Ch>[:STATe] ON | OFF
Create or delete diagram area	DISPlay:WINDow<Wnd>:STATe ON | OFF
Display trace in diagram area	DISPlay:WINDow<Wnd>:TRACe<WndTr>:FEED

The assignment between traces, channels, and diagram areas is defined via numeric suffixes as illustrated in the following example:

Example:

```
CALC4:PAR:SDEF 'Ch4Tr1', 'S11'
```

Create channel 4 (channel suffix 4) and a trace named "Ch4Tr1" to measure the input reflection coefficient S11. The trace is created but not displayed.

```
DISP:WIND2:STAT ON
```

Create diagram area no. 2 (window suffix 2).

```
DISP:WIND2:TRAC9:FEED 'CH4TR1'
```

Display the generated trace (identified by its name "Ch4Tr1") in diagram area no. 2 (window suffix 2), assigning the trace number 9 (trace suffix 9) to it.

6.3.2 Active traces in remote control

In manual control there is always exactly one active trace, irrespective of the number of channels and traces defined. The "active channel" contains the active trace; see Section 4.1.3.1, "Trace settings", on page 81.

In remote control, each channel contains an active trace (unless the channel contains no trace at all). This principle actually simplifies the remote control command syntax, because it allows the active trace in a particular channel to be referenced by the channel suffix. No additional trace identifier is needed; there is no need either to distinguish channel and trace settings using mnemonics or suffixes.

The active traces are handled as follows:

- After a preset (*RST), the analyzer displays a single diagram area with the default trace no. 1 named Trc1. The trace is active in manual and in remote control.
- In manual control, a new, added trace automatically becomes the active trace. To select another trace as the active trace, tap inside the trace list.
- In remote control, a new trace added via CALCulate<Ch>:PARameter:SDEFine '<trace_name>', '<parameter>' also becomes the active trace. To select another trace as the active trace, use (CALCulate<Ch>:PARameter:SELect '<trace_name>').

- The active traces for manual and remote control can be different.

The following program example illustrates how to create, select and reference traces. It is instructive to observe the analyzer screen to check the effect of each step.

Example:

```
*RST
```

Reset the analyzer, creating channel no. 1 with the default trace "Trc1". The trace is displayed in diagram area no. 1.

```
CALC1:PAR:SDEF 'Trc2', 'S11';DISP:WIND:TRAC2:FEED 'Trc2'
```

Create a trace named "Trc2", assigned to channel no. 1 (the suffix 1 after `CALC` can be omitted), and display the trace. The new trace automatically becomes the active trace for manual and for remote control. To check this, tap "Trace – Marker – Marker 1" to create a marker. The marker is assigned to "Trc2". Delete all markers ("Trace – Marker – All Markers Off").

```
CALC1:MARK ON
```

Example:

To verify that "Trc2" is also active for remote control, use the channel suffix 1 after `CALC` (can be omitted) to reference the active trace in channel 1 and create a marker "Mkr 1". The marker is assigned to "Trc2".

Example:

```
CALC:PAR:SEL 'Trc1';CALC1:MARK ON
```

Select the old default trace "Trc1" as the active trace for remote control. Create a marker to verify that "Trc1" is now the active trace in channel 1.



In the SCPI command description, the numeric suffix `<Ch>` is used for channel settings (it denotes the configured channel), whereas `<Chn>` is used for trace settings (it refers to the active trace in the channel).

6.3.3 Initiating measurements, speed considerations

After a reset the network analyzer measures in continuous mode. The displayed trace shows the result of the last sweep and is continuously updated. This provides a permanent visual control over the measurement and the effect of any analyzer settings.

In remote control, it is advisable to follow a different approach to use the analyzer's resources to full capacity and gain measurement speed. The following principles can help to optimize a remote control program (see also programming example in Section 8.1.1, "Typical stages of a remote control program", on page 1647):

- Switch off the measurement while configuring your instrument.
- Use a minimum number of suitably positioned sweep points.
- Start a single sweep, observing proper command synchronization, and retrieve your results.

The following command sequence performs a single sweep in a single channel.

Example:

```
*RST; :INITiate:CONTinuous:ALL OFF
```

Activate single sweep mode for all channels (including the channels created later).

```
INITiate1:IMMediate; *WAI
```

Start a single sweep in channel no. 1, wait until the sweep is terminated before proceeding to the next command (see Section 6.4, "Command processing", on page 886).

Sweeps in several channels

It is also possible to subdivide the channels within a recall set into active and inactive channels. The analyzer only measures the active channels; see the programming example for `CONFigure:CHANnel<Ch>:MEASure[:STATe]`.

6.3.4 Addressing traces and channels

The analyzer provides various schemes for addressing traces and channels and for querying trace and channel names. The following tables give an overview.

Table 6-2: Addressing channels

Method	Commands / Example
Channel number <Ch> as a numeric suffix	`CONFigure:CHANnel<Ch>[:STATe] ON`
Query all channel names	`CONFigure:CHANnel:CATalog?` (returns the names of all channels)
Assign or query channel name of a channel numbered <Ch>	`CONFigure:CHANnel<Ch>:NAME 'ABCD'` `CONFigure:CHANnel<Ch>:NAME?` (returns 'ABCD')
Query channel number assigned to a channel named 'ABCD'	`CONFigure:CHANnel<Ch>:NAME:ID? 'ABCD'` (returns the actual channel number, the channel suffix is ignored)

Table 6-3: Addressing traces

Method	Commands / Example
Channel number <Chn> used as a reference for the active trace in the channel	`CALCulate<Chn>:MARKer<Mk>[:STATe] ON`
Trace name (string variable) used as a reference for the trace	`CALCulate<Ch>:PARameter:DELete '<Trace Name>'`
Trace number <Trc> as a numeric suffix (exception!)	`CONFigure:TRACe<Trc>:NAME?`
Trace number <WndTr> within a particular diagram area <Wnd>	`DISPlay:WINDow<Wnd>:TRACe<WndTr>:FEED`
Query all trace names	`CONFigure:TRACe:CATalog?` (returns the names of all traces)

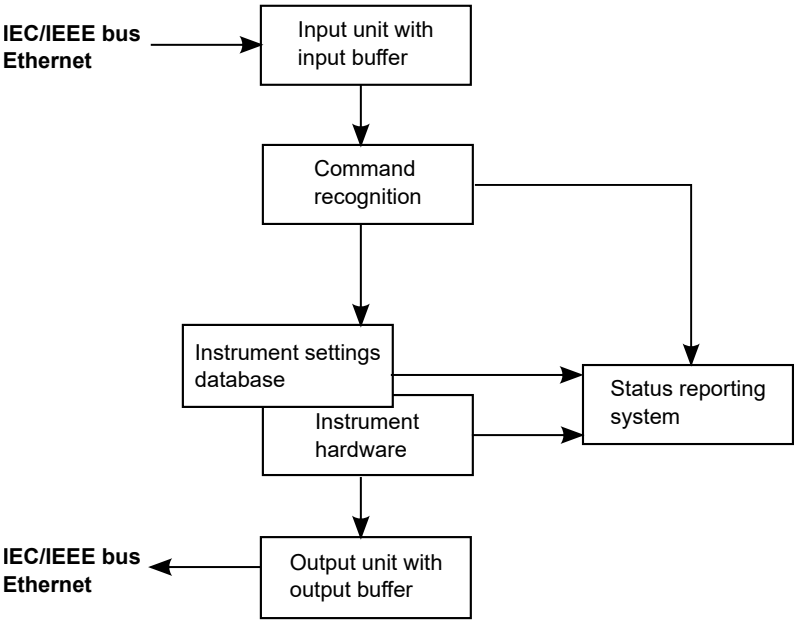
Method	Commands / Example
Assign or query trace name of a trace numbered <Trc>	`CONFigure:TRACe<Trc>:NAME 'ABCD'` `CONFigure:TRACe<Trc>:NAME?` (returns `'ABCD'`)
Query trace number assigned to a trace named 'ABCD'	`CONFigure:TRACe<Trc>:NAME:ID? 'ABCD'` (returns the actual trace number; the trace suffix is ignored)

Table 6-4: Mixed commands

Method	Commands / Example
Query channel name for a trace referenced by its trace name	`CONFigure:TRACe<Trc>:CHANnel:NAME?` 'ABCD' (returns the channel name for trace `'ABCD'`; the trace suffix is ignored)
Query channel number for a trace referenced by its trace name	`CONFigure:TRACe<Trc>:CHANnel:NAME:ID? 'ABCD'` (returns the actual channel number for trace `'ABCD'`; the trace suffix is ignored)

6.4 Command processing

The block diagram below shows how commands are serviced in the instrument.



The individual components work independently and simultaneously. They communicate with each other with so-called messages.

6.4.1 Input unit

The input unit receives commands character by character from the controller and collects them in the input buffer. The input unit sends a message to the command recog-

nition when the input buffer is full or when it receives a delimiter, <PROGRAM MESSAGE TERMINATOR>, as defined in IEEE 488.2, or the interface message DCL.

If the input buffer is full, the message data traffic is stopped and the data received up to then is processed. Subsequently the traffic is continued. If, however, the buffer is not yet full when receiving the delimiter, the input unit can already receive the next command during command recognition and execution. The receipt of a DCL clears the input buffer and immediately initiates a message to the command recognition.

6.4.2 Command recognition

The command recognition stage analyzes the data received from the input unit. It proceeds in the order in which it receives the data. Only a DCL is serviced with priority, e.g. a GET (Group Execute Trigger) is only executed after the commands received before. Each recognized command is immediately transferred to the data set but not executed immediately.

The command recognition detects syntax errors in the commands and transfers them to the status reporting system. The rest of a command line after a syntax error is still executed, if possible. After the syntax check, the range of the numerical parameters is checked, if necessary.

If the command recognition detects a delimiter or a DCL, it also requests the data set to perform the necessary instrument hardware settings. Subsequently it is immediately prepared to process further commands. This means that new commands can already be serviced while the hardware is still being set (overlapping execution).

6.4.3 Data base and instrument hardware

The expression instrument hardware denotes the part of the instrument fulfilling the actual instrument function – signal generation, measurement etc. The controller is not included. The data base manages all the parameters and associated settings required for the instrument hardware.

Setting commands lead to an alteration in the data set. The data set management enters the new values (e.g. frequency) into the data set, however, it only passes them on to the hardware when requested by the command recognition. This can only occur at the end of a command line, therefore the order of the setting commands in the command line is not relevant.

The commands are only checked for their compatibility among each other and with the instrument hardware immediately before they are transmitted to the instrument hardware. If the instrument detects that execution is not possible, an execution error is signaled to the status reporting system. All alterations of the data set are canceled, the instrument hardware is not reset. Due to the delayed checking and hardware setting, however, impermissible instrument states can be set for a short period of time within one command line without this leading to an error message (example: simultaneous activation of a frequency and a power sweep). At the end of the command line, however, a permissible instrument state must have been reached again.

Before passing on the data to the hardware, the settling bit in the `STATus:OPERation` register is set (see Section 6.5.3.4, "STATus:OPERation", on page 897). The hardware executes the settings and resets the bit again when the new state has settled. This fact can be used to synchronize command servicing.

Queries induce the data set management to send the desired data to the output unit.

6.4.4 Status reporting system

The status reporting system collects information on the instrument state and makes it available to the output unit on request. The exact structure and function are described in Section 6.5, "Status reporting system", on page 890.

6.4.5 Output unit

The output unit collects the information requested by the controller, which it receives from the data set management. It processes it according to the SCPI rules and makes it available in the output buffer. If the information requested is longer, it is made available in portions without this being recognized by the controller.

If the instrument is addressed as a talker without the output buffer containing data or awaiting data from the data set management, the output unit sends the error message `Query UNTERMINATED` to the status reporting system. No data is sent on the GPIB bus or via the Ethernet, the controller waits until it has reached its time limit. This behavior is specified by SCPI.

6.4.6 Command sequence and command synchronization

IEEE 488.2 defines a distinction between overlapped and sequential commands:

- A sequential command is one which finishes executing before the next command starts executing. Commands that are processed quickly are usually implemented as sequential commands.
- An overlapping command is one which does not automatically finish executing before the next command starts executing. Usually, overlapping commands take longer to process and allow the program to do other tasks while being executed. If overlapping commands do have to be executed in a defined order, e.g. to avoid wrong measurement results, they must be serviced sequentially. This is called synchronization between the controller and the analyzer.

According to Section 6.4.3, "Data base and instrument hardware", on page 887, setting commands within one command line, even though they may be implemented as sequential commands, are not necessarily serviced in the order in which they have been received. To make sure that commands are carried out in a certain order, each command must be sent in a separate command line.

Example: Commands and queries in one message

The response to a query combined in a program message with commands that affect the queried value is not predictable. Sending

```
:FREQ:STAR 1GHZ;SPAN 100
:FREQ:STAR?
```

always returns `1000000000` (1 GHz). When:

```
:FREQ:STAR 1GHz;STAR?;SPAN 1000000
```

is sent, however, the result is not specified by SCPI. The result could be the value of `STARt` before the command was sent since the instrument can defer executing the individual commands until a program message terminator is received. The result could also be 1 GHz if the instrument executes commands as they are received.

As a rule, send commands and queries in different program messages.

Example: Overlapping command with *OPC

The analyzer implements `INITiate[:IMMediate]...` commands as overlapped commands. Assuming, e.g., that `INITiate[:IMMediate][:DUMMy]` takes longer to execute than `*OPC`, sending the command sequence:

```
INIT; *OPC.
```

results in initiating a sweep and, after some time, setting the OPC bit in the ESR. Sending the commands:

```
INIT; *OPC; *CLS
```

still initiates a sweep. Since the operation is still pending when the analyzer executes *CLS, forcing it into the Operation Complete Command Idle State (OCIS), *OPC is effectively skipped. The OPC bit is not set until the analyzer executes another *OPC command.



The analyzer provides only two overlapped commands, `INITiate<Ch>[:IMMediate][:DUMMy]` and `INITiate<Ch>[:IMMediate]:ALL`. What is said below is not relevant for the other (sequential) SCPI commands.



Preventing overlapping execution

To prevent an overlapping execution of commands, one of the commands ***OPC**, `*OPC?` or `*WAI` can be used. For a programming example, refer to Section 8.1.1.3, "Start of the measurement and command synchronization", on page 1648.

Command	Action after the hardware has settled	Programming the controller
*WAI	Stops further command processing until all commands sent before *WAI have been executed Note: The GPIB bus handshake is not stopped	Send *WAI directly after the command which should be terminated before the next command is executed.
*OPC?	Stops command processing until 1 is returned, i.e. until the "Operation Complete" bit has been set in the ESR. This bit indicates that the previous commands have been completed.	Send *OPC? directly after the command which should be terminated before the next command is executed.
*OPC	Sets the operation complete bit in the ESR after all previous commands have been executed.	– Set bit 0 in the ESE – Set bit 5 in the SRE – Wait for service request (SRQ)

Example: Non-blocking synchronization on overlapping command termination

*CLS resets the ESR.

INITiate1:IMMediate; OPC* starts a single sweep in channel 1 and sets the OPC bit (bit 0) of the ESR when the sweep has finished. It does not stop further command processing.

With *ESR?, you can check for the value of the status bit to see if the sweep has finished.

6.5 Status reporting system

The status reporting system stores all information on the present operating state of the instrument, and on errors which have occurred. This information is stored in the status registers and in the error queue. Both can be queried using the STATus... commands.

Hierarchy of status registers

As shown in section Overview of status registers, the status information is of hierarchical structure.

- STB, SRE:
  The STatus Byte (STB) register and its associated mask register Service Request Enable (SRE) make up the highest level of the status reporting system. The STB provides a rough overview of the instrument status, collecting the information of the lower-level registers.
- The STB receives its information from:
  The Event Status Register (ESR) with the associated mask register standard Event Status Enable (ESE).
  The STATus:OPERation and STATus:QUEStionable registers which are defined by SCPI and contain detailed information on the instrument.
- IST, PPE:
  The Individual STatus (IST) flag, like the SRQ, combines the entire instrument status in a single bit. The PPE is associated to the IST flag. It fulfills an analogous function for the IST flag as the SRE does for the service request.

- Output buffer:
  Contains the messages that the instrument returns to the controller. It is not part of the status reporting system but determines the value of the MAV bit in the STB.

All status registers have the same internal structure, see Structure of a SCPI status register.

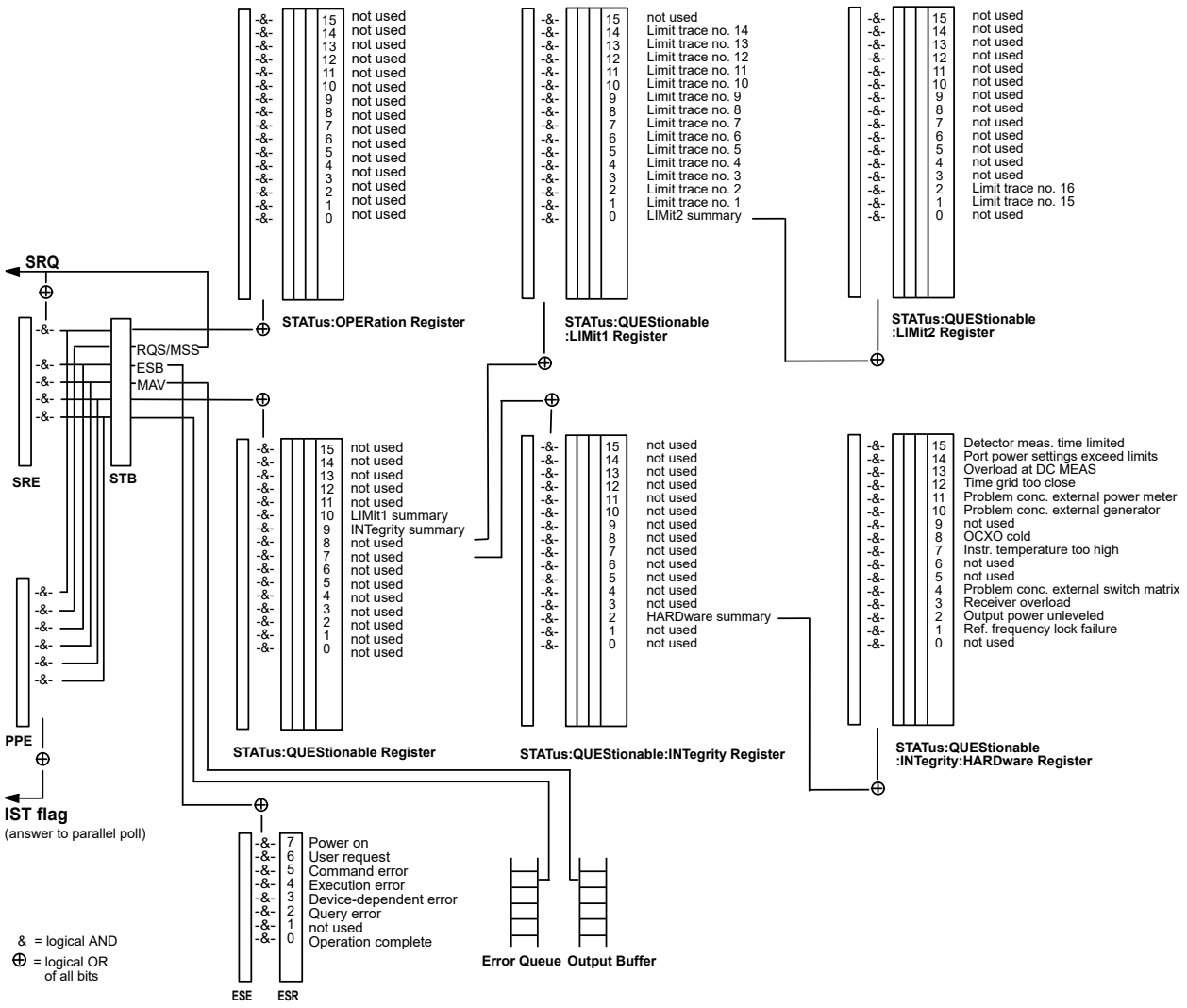
For more information on the individual status registers, see Contents of the status registers.

SRE register

The service request enable register SRE can be used as ENABle part of the STB if the STB is structured according to SCPI. By analogy, the ESE can be used as the ENABle part of the ESR.

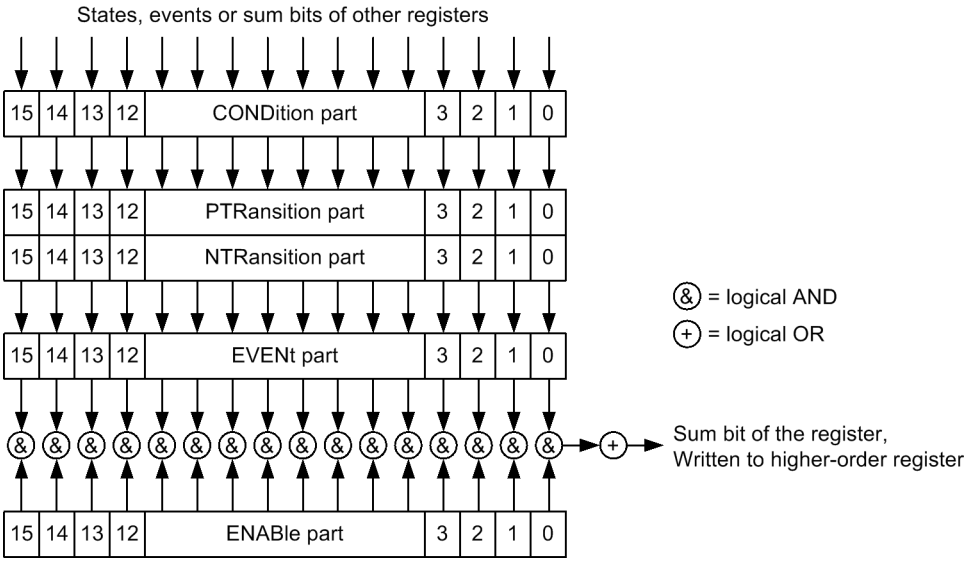
6.5.1 Overview of status registers

The status registers of the R&S ZNB are implemented as shown below.



6.5.2 Structure of a SCPI status register

Each standard SCPI register consists of 5 parts which each have a width of 16 bits and have different functions. The individual bits are independent of each other, i.e. each hardware status is assigned a bit number which is valid for all five parts. Bit 15 (the most significant bit) is set to zero for all parts. Thus the contents of the register parts can be processed by the controller as positive integer.



The sum bit is obtained from the EVENt and ENABle part for each register. The result is then entered into a bit of the CONDition part of the higher-order register.

The instrument automatically generates the sum bit for each register. Thus an event can lead to a Service request throughout all levels of the hierarchy.

The five parts of a SCPI register have different properties and function as described below.

CONDition

The CONDition part is permanently overwritten by the hardware or the sum bit of the next lower register. Its contents always reflect the current instrument state.

This register part can only be read, but not overwritten or cleared. Reading the CONDition register is nondestructive.

PTRansition

The two transition register parts define which state transition of the condition part (none, 0 to 1, 1 to 0 or both) is stored in the EVENt part.

The Positive TRansition part acts as a transition filter. When a bit of the CONDition part is changed from 0 to 1, the associated PTR bit decides whether the EVENt bit is set to 1:

- PTR bit = 1: the EVENt bit is set
- PTR bit = 0: the EVENt bit is not set

This status register part can be overwritten and read at will. Reading the PTRansition register is nondestructive.

NTRansition

The Negative TRansition part also acts as a transition filter. When a bit of the CONDition part is changed from 1 to 0, the associated NTR bit decides whether the EVENt bit is set to 1.

- NTR bit = 1: the EVENt bit is set.
- NTR bit = 0: the EVENt bit is not set.

This part can be overwritten and read at will. Reading the PTRansition register is non-destructive.

EVENt

The EVENt part indicates whether an event has occurred since the last reading, it is the "memory" of the condition part. It only indicates events passed on by the transition filters. It is permanently updated by the instrument. This part can only be read by the user. Reading the register clears it. This part is often equated with the entire register.

ENABle

The ENABle part determines whether the associated EVENt bit contributes to the sum bit (cf. below). Each bit of the EVENt part is ANDed with the associated ENABle bit (symbol '&'). The results of all logical operations of this part are passed on to the sum bit via an OR function (symbol '+').

- ENAB bit = 0: The associated EVENt bit does not contribute to the sum bit.
- ENAB bit = 1: If the associated EVENT bit is "1", the sum bit is set to "1" as well.

This part can be overwritten and read by the user at will. Its contents are not affected by reading.

The **sum bit** is obtained from the EVENt and ENABle part for each register. The result is then entered into a bit of the CONDition part of the higher-order register. The instrument automatically generates the sum bit for each register. Thus an event can lead to a service request throughout all levels of the hierarchy.

6.5.3 Contents of the status registers

The individual status registers are used to report different classes of instrument states or errors. The following status registers belong to the general model described in IEEE 488.2:

- The STatus Byte (STB) gives a rough overview of the instrument status.
- The Individual STatus (IST) flag combines the entire status information into a single bit that can be queried in a Parallel poll.
- The Event Status Register (ESR) indicates general instrument states.

The status registers below belong to the device-dependent SCPI register model:

- The `STATus:OPERation` register contains conditions which are part of the instrument's normal operation.
- The `STATus:QUEStionable` register indicates whether the data currently being acquired is of questionable quality.

- The `STATus:QUEStionable:LIMit<1|2>` register indicates the result of the limit check.
- The `STATus:QUEStionable:INTegrity` register monitors hardware failures of the analyzer.

6.5.3.1 STB and SRE

The STatus Byte (STB) provides a rough overview of the instrument status by collecting the pieces of information of the lower registers. The STB represents the highest level within the SCPI hierarchy. A special feature is that bit 6 acts as the summary bit of the remaining bits of the status byte.

The STB is linked to the Service Request Enable (SRE) register on a bit-by-bit basis.

- The STB corresponds to the EVENt part of a SCPI register, indicating the current instrument state. This register is cleared when it is read.
- The SRE corresponds to the ENABle part of a SCPI register. If a bit is set in the SRE and the associated bit in the STB changes from 0 to 1, a Service request (SRQ) is generated. Bit 6 of the SRE is ignored, because it corresponds to the summary bit of the STB.

The bits in the STB are defined as follows:

Bit No.	Meaning
2	**Error queue not empty** If this bit is enabled by the SRE, each entry of the error queue generates a Service request (SRQ). Thus an error can be recognized and further pinned down by polling the error queue. The poll provides an informative error message.
3	**QUEStionable status summary bit** This bit is set if an EVENt bit is set in the STATus:QUEStionable register and the associated ENABle bit is set to 1. The bit indicates a questionable instrument status, which can be further pinned down by polling the QUEStionable register.
4	**MAV bit** (message available) This bit is set if a message is available and can be read from the output buffer. This bit can be used to transfer data from the instrument to the controller automatically.
5	**ESB bit** Sum bit of the event status register. It is set if one of the bits in the event status register is set and enabled in the event status enable register. Setting of this bit implies an error or an event which can be further pinned down by polling the event status register.
6	**MSS bit** (master status summary bit) This bit is set if the instrument triggers a service request, which happens if one of the other bits of this register is set together with its mask bit in the SRE register.
7	**OPERation status register summary bit** This bit is set if an EVENt bit is set in the OPERation status register and the associated ENABle bit is set to 1. The bit indicates that the instrument is performing an action. The type of action can be determined by polling the STATus:OPERation register.

Related common commands

The STB is read out using the command `*STB?` or a Serial poll.

The SRE can be set using command `*SRE` and read using `*SRE?`.

6.5.3.2 IST flag and PPE

In analogy to the Service request (SRQ), the Individual STatus (IST) flag combines the entire status information in a single bit. It can be queried by a Parallel poll.

The Parallel Poll Enable (PPE) register determines which bits of the STB contribute to the IST flag. The bits of the STB are ANDed with the corresponding bits of the PPE, with bit 6 being used as well in contrast to the SRE. The IST flag results from ORing all results.

Related common commands

The IST flag is queried using the common command `*IST?`. The PPE can be set using `*PRE` and read using `*PRE?`.

See also Common commands.

6.5.3.3 ESR and ESE

The Event Status Register (ESR) indicates general instrument states. It is linked to the Event Status Enable (ESE) register on a bit-by-bit basis.

- The ESR corresponds to the CONDition part of a SCPI register indicating the current instrument state (although reading is destructive).
- The ESE corresponds to the ENABle part of a SCPI register. If a bit is set in the ESE and the associated bit in the ESR changes from 0 to 1, the ESB bit in the STatus Byte (STB) is set.

The bits in the ESR are defined as follows:

Bit No.	Meaning
0	**Operation complete** This bit is set on receipt of the command *OPC after all previous commands have been executed.
2	**Query error** This bit is set if either the controller wants to read data from the instrument without having sent a query, or if it does not fetch requested data and sends new instructions to the instrument instead. The cause is often a query which is faulty and hence cannot be executed.
3	**Device-dependent error** This bit is set if a device-dependent error occurs. An error message with a number between –300 and –399 or a positive error number, which describes the error in greater detail, is entered into the error queue. See also Section 9, "Error messages and troubleshooting", on page 1687.

Bit No.	Meaning
4	**Execution error** This bit is set if a received command is syntactically correct, but cannot be performed for other reasons. An error message with a number between –200 and –300, which describes the error in greater detail, is entered into the error queue.
5	**Command error** This bit is set if a command which is undefined or syntactically incorrect is received. An error message with a number between -100 and -200, which describes the error in greater detail, is entered into the error queue.
6	**User request** This bit is set when the instrument is switched over to manual control or when a user-defined softkey is used (see `SYSTem:USER:KEY`).
7	**Power On** (supply voltage on) This bit is set when the instrument is switched on.

Related common commands

The Event Status Register (ESR) can be queried using `ESR?`. The Event Status Enable (ESE) register can be set using the command `*ESE` and read using `*ESE?`.

See also Common commands.

6.5.3.4 STATus:OPERation

The STATus:OPERation register contains conditions which are part of the instrument's normal operation. The analyzer does not use the STATus:OPERation register.

6.5.3.5 STATus:QUEStionable

The `STATus:QUEStionable` register indicates whether the acquired data is of questionable quality and monitors hardware failures of the analyzer. It can be queried using the commands `STATus:QUEStionable:CONDition?` or `STATus:QUEStionable:EVENt?`.

Bit No.	Meaning
9	**INTegrity register summary** This bit is set if a bit is set in the STATus:QUEStionable:INTegrity register and the associated ENABle bit is set to 1.
10	**LIMit register summary** This bit is set if a bit is set in the STATus:QUEStionable:LIMit1 register and the associated ENABle bit is set to 1.

STATus:QUEStionable:LIMit<1|2>

The `STATus:QUEStionable:LIMit<1|2>` registers indicate the result of the limit check. They can be queried using the commands `STATus:QUEStionable:LIMit<1|2>:CONDition?` or

STATus:QUEStionable:LIMit<1|2>[:EVENt]?
STATus:QUEStionable:LIMit1 is also the summary register of the lower level STATus:QUEStionable:LIMit2 register.

The bits in the STATus:QUEStionable:LIMit1 register are defined as follows:

Bit No.	Meaning
0	**LIMit2 register summary** This bit is set if a bit is set in the STATus:QUEStionable:LIMit2 register and the associated ENABle bit is set to 1.
1	**Failed limit check for trace no. 1** This bit is set if any point on trace no. 1 fails the limit check.
...	...
14	**Failed limit check for trace no. 14** This bit is set if any point on trace no. 14 fails the limit check.

The bits in the STATus:QUEStionable:LIMit2 register are defined as follows:

Bit No.	Meaning
0	Not used
1	**Failed limit check for trace no. 15** This bit is set if any point on trace no. 15 fails the limit check.
2	**Failed limit check for trace no. 16** This bit is set if any point on trace no. 16 fails the limit check.

Numbering of traces

The traces numbers 1 to 16 are assigned as follows:

- Traces assigned to channels with smaller channel numbers have smaller trace numbers.
- Within a channel, the order of traces reflects their creation time: The oldest trace has the smallest, the newest trace has the largest trace number. This is equivalent to the order of traces in the response string of the CALCulate<Ch>:PARameter:CATalog? query.
- The number of traces monitored cannot exceed 16. If a setup contains more traces, the newest traces are not monitored.

STATus:QUEStionable:INTegrity...

The STATus:QUEStionable:INTegrity register monitors hardware failures of the analyzer. It can be queried using the commands
STATus:QUEStionable:INTegrity:CONDition? or
STATus:QUEStionable:INTegrity[:EVENt]?
STATus:QUEStionable:INTegrity is also the summary register of the lower level STATus:QUEStionable:INTegrity:HARDware register.

Refer to the Section 9, "Error messages and troubleshooting", on page 1687 for a detailed description of hardware errors including possible remedies.

The bits in the `STATus:QUEStionable:INTegrity` register are defined as follows.

Bit No.	Meaning
2	**HARDware register summary** This bit is set if a bit is set in the STATus:QUEStionable:INTegrity:HARDware register and the associated ENABle bit is set to 1.

STATus:QUEStionable:INTegrity:HARDware

The `STATus:QUEStionable:INTegrity:HARDware` register can be queried using the commands `STATus:QUEStionable:INTegrity:HARDware:CONDition?` or `STATus:QUEStionable:INTegrity:HARDware[:EVENt]?`

The bits in the `STATus:QUEStionable:INTegrity:HARDware` register are defined as follows.

Bit No.	Meaning
0	Not used
1	**Reference frequency lock failure** If an external reference signal or an internal high precision clock (option B4) is used, the local oscillator is phase locked to a reference signal. This bit is set if this phase-locked loop (PLL) fails. For external reference: check frequency and level of the supplied reference signal.
2	**Output power unleveled** This bit is set if the level control at one of the ports is unsettled or unstable, possibly due to an external disturbing signal. Change generator level at the port; check external components.
3	**Receiver overload protection tripped** This bit is set if the analyzer detects an excessive input level at one of the ports. If this condition persists, all internal and external generators are switched off. Reduce RF input level at the port. Check amplifiers in the external test setup, then switch on the internal source using `OUTPut ON`.
4	**Problem concerning external switch matrix** This bit is set if an external matrix has been configured but cannot be controlled or provides error messages. Check whether the matrix is properly connected and switched on. Check for proper wiring of the interfaces, in particular on input and output. If the LAN or USB interface is configured, disconnect the Direct CTRL plug. Exclude address conflicts when using several external switch matrices or other external devices.
6	**Internal communication error** This bit is set if an internal error caused the analyzer to perform an automatic hardware reset. The current measurement results are possibly invalid. The bit is automatically cleared at the beginning of the next sweep, no action is required.

Bit No.	Meaning
7	**Instrument temperature is too high** This bit is set if the analyzer detects that the instrument temperature is too high. Reduce ambient temperature, keep ventilation holes of the casing unobstructed.
8	**OCXO cold** This bit is set if the oven for the internal high precision clock (option B4) is not at its operating temperature. Wait until the oven has been heated up.
9	**Unstable level control** This bit is set if the analyzer detects an excessive source level at one of the ports. The signal is turned off and the sweep halted. Check signal path for the received wave, especially check external components. Then restart the sweep (`INITiate<Ch>[:IMMediate]`).
10	**Problem concerning external generator** This bit is set if an external generator has been configured but cannot be controlled or provides error messages. Check whether the generator is properly connected and switched on. Check the GPIB address; exclude address conflicts when using several external generators or other equipment.
11	**Problem concerning external power meters or DC measurement devices** This bit is set if an external power meter or DC measurement device has been configured, but cannot be controlled or provides error messages. Check whether the device is properly connected and switched on. Check its address and resolve address conflicts (with external devices of any type).
12	**Time grid too close** This bit is set if the sweep points for a time sweep are too close, so that the analyzer cannot process the measurement data until the next sweep point starts. Increase stop time, reduce no. of points, increase IF bandwidth. If possible reduce number of partial measurements, e.g. by restricting the number of ports measured.
13	**Overload at DC MEAS** This bit is set if the input voltage at one of the DC input connectors on the rear panel is too high. Reduce the input voltage.
14	**Power settings exceed hardware limits** This bit is set if the source power at one of the test ports is too high or too low. Reduce or increase the source power.
15	**Detector MEAS time has been internally limited** This bit is set if the selected measurement time for a detector (observation time) is too long. If desired, reduce the measurement time or select a smaller IF bandwidth.

6.5.4 Application of the status reporting system

The purpose of the status reporting system is to monitor the status of one or several devices in a measuring system. To do this and react appropriately, the controller must

receive and evaluate the information of all devices. The following standard methods described in the following sections are used:

- Service request (SRQ) initiated by the measuring device
- Serial poll of all devices in the bus system, initiated by the controller to find out who sent an SRQ and why
- Parallel poll of all devices
- Query of a specific instrument status by commands
- Query of the error queue

6.5.4.1 Service request

The R&S ZNB can send a service request (SRQ) to the controller. Usually this service request causes an interrupt, to which the control program can react appropriately.

Initiating an SRQ

As shown in section Overview of status registers, an SRQ is initiated if one or several of bits 2, 3, 4, 5 or 7 of the status byte are set and enabled in the SRE. Each of these bits summarizes the information of a further register, the error queue or the output buffer.

The ENABle parts of the status registers can be set such that arbitrary bits in an arbitrary status register initiate an SRQ. To use the possibilities of the service request effectively, all bits in the enable registers SRE and ESE must be set to "1".

Example: Use *OPC to generate an SRQ

1. Set bit 0 in the ESE (Operation Complete).
2. Set bit 5 in the SRE (ESB).
3. Insert `*OPC` in the command sequence (e.g. at the end of a sweep).

When all commands preceding `*OPC` have been completed, the instrument generates an SRQ.

Example: Generate an SRQ when a limit is exceeded

1. Set bit 3 in the SRE (summary bit of the `STATus:QUEStionable` register, set after `STATus:PRESet`)
2. Set bit 10 in the `STATus:QUEStionable:ENABle` register (summary bit of the `STATus:QUEStionable:LIMit1` register)
3. Set bit 1 in the `STATus:QUEStionable:LIMit1:ENABle` register

The R&S ZNB generates an SRQ when the event associated with bit 1 of the `STATus:QUEStionable:LIMit1:ENABle` register occurs, i.e. when any point on the first trace fails the limit check.

Example: Find out which event caused an SRQ

The procedure to find out which event caused an SRQ is analogous the procedure to generate an SRQ:

1. `STB?` (query the contents of the status byte in decimal form)
   If bit 3 (QUEStionable summary bit) is set, then:
2. `STAT:QUES:EVENT?` (query STATus:QUEStionable register)
   If bit 10 (QUEStionable:LIMit1 summary bit) is set, then:
3. Query `STAT:QUES:LIMit1:EVENT?` (query `STATus:QUEStionable:LIMit1` register)
   If bit 1 is set, then the first trace failed the limit check.

The SRQ is the only possibility for the instrument to become active on its own. Each controller program should set the instrument such that a service request is initiated in the case of malfunction. The program should react appropriately to the service request.

6.5.4.2 Serial poll

In a serial poll, the controller queries the STatus Bytes of the devices in the bus system one after another. The query is made via interface messages, so it is faster than a poll using `*STB?`.

The serial poll method is defined in IEEE 488.1 and used to be the only standard possibility for different instruments to poll the status byte. The method also works for instruments which do not adhere to SCPI or IEEE 488.2.

The serial poll is used to obtain a fast overview of the state of several instruments connected to the controller.

6.5.4.3 Parallel poll

In a parallel poll, up to eight instruments are simultaneously requested by the controller with a single command to transmit 1 bit of information each on the data lines, i.e., to set the data line allocated to each instrument to a logical "0" or "1".

In addition to the SRE register, which determines the conditions under which an SRQ is generated, there is a Parallel Poll Enable (PPE) register. This register is ANDed with the STB bit by bit, considering bit 6 as well. The results are ORed, the result is possibly inverted and then sent as a response to the parallel poll of the controller. The result can also be queried without parallel poll with the command `*IST?`.

The parallel poll method is used to find out quickly which one of the instruments connected to the controller has sent a service request. To this effect, SRE and PPE must be set to the same value.

6.5.4.4 Query of an instrument status

Each part of any status register can be read using queries. There are two types of commands:

- The common commands `*ESR?, *IDN?, *IST?, *STB?` query the higher-level registers.
- The commands of the STATus system query the SCPI registers (e.g. `STATus:OPERation...`)

All queries return a decimal number which represents the bit pattern of the status register. This number is evaluated by the controller program.

Queries are usually used after an SRQ to obtain more detailed information on its cause.

Decimal representation of a bit pattern

The STB and ESR registers contain 8 bits, the SCPI registers 16 bits. The contents of a status register is keyed and transferred as a single decimal number. To make this possible, each bit is assigned a weighted value. The decimal number is calculated as the sum of the weighted values of all bits in the register that are set to 1.

Bits	0	1	2	3	4	5	6	7	...
Weight	1	2	4	8	16	32	64	128	...

Example: The decimal value 40 = 32 + 8 indicates that bits no. 3 and 5 in the status register (e.g. the QUEStionable status summary bit and the ESB bit in the STB) are set.

6.5.4.5 Error queue

Each error state in the instrument leads to an entry in the error queue, whose current number of entries can be queried using `SYSTem:ERRor:COUNt?`. The entries of the error queue are detailed plain text error messages that can be queried via remote control using `SYSTem:ERRor[:NEXT]?` or `SYSTem:ERRor:ALL?`. Each call of `SYSTem:ERRor[:NEXT]?` provides one entry from the error queue. If no error messages are stored there anymore, the instrument responds with `0, "No error"`.

The error queue should be queried after every SRQ in the controller program as the entries describe the cause of an error more precisely than the status registers. Especially in the test phase of a controller program the error queue should be queried regularly since faulty commands from the controller to the instrument are recorded there as well.

6.5.5 Reset values of the status reporting system

The table below indicates the effects of various commands upon the status reporting system of the R&S ZNB.

DCL, SDC (Device Clear, Selected Device Clear)

Event	Switching on supply voltage Power-On-Status-Clear		DCL, SDC (Device Clear, Selected Device Clear)	*RST or SYSTem:PRESet:ALL	STATus:PRESet	*CLS
Effect	0	1				
Clear STB, ESR		yes				yes
Clear SRE, ESE		yes				
Clear PPE		yes				
Clear EVENt parts of the registers		yes				yes
Clear ENABle parts of all OPERation and QUESTionable registers, Fill ENABle parts of all other registers with "1".		yes			yes	
Fill PTRansition parts with "1" Clear NTRansition parts		yes			yes	
Clear error queue	yes	yes				yes
Clear output buffer	yes	yes	yes	1)	1)	1)
Clear command processing and input buffer	yes	yes	yes			

1) Every command being the first in a command line, i.e. immediately following a <PROGRAM MESSAGE TERMINATOR> clears the output buffer.

6.6 Automating tasks with remote command scripts

To configure a test setup quickly and make complex test setups or repetitive measurements reproducible, you can automate the required settings with scripts. A script contains a series of SCPI commands corresponding to the settings. When completed, it is converted to an executable format, saved in a file, and can be run whenever needed.

Creating a SCPI script

Using the SCPI Recorder functions, you can create a SCPI script directly on the instrument and then export the script for use on the controller. You can also edit or write a script manually, using a suitable editor on the controller. For manual creation, the instrument supports you by showing the corresponding command syntax for the current setting value.

You can create a SCPI script directly on the instrument at any time of operation, as follows:

- Recording all performed steps automatically

The instrument records the SCPI command and settings value of each step you perform, and then writes the commands to the file system, see "How to record SCPI commands automatically" on page 908. You can start, stop and resume automatic recording of SCPI commands.

6.6.1 The SCPI recorder



Access: Toolbar

The "SCPI Recorder" displays a list of the currently recorded commands and provides functions to create and export a script of SCPI commands. It also displays the context sensitive help for each SCPI command in a separate window.

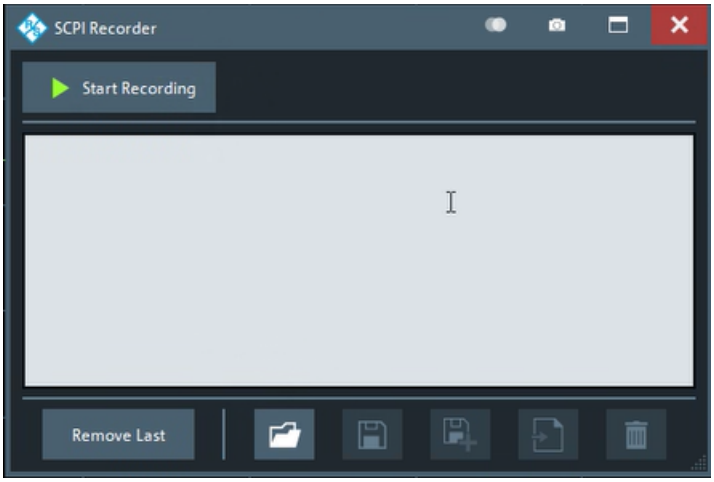


Figure 6-1: SCPI Recorder

6.6.1.1 Recording

Start Recording....905
List of recorded commands/script editor....906
Remove Last....906
Load Recording....906
Save Recording....906
Save As Recording....906
Export As....907
Clear All....907
Help....907

Start Recording

If clicked, the SCPI Recorder automatically records the required SCPI commands and parameter values for the settings and functions you use while operating the R&S ZNB.

To view the list of currently recorded SCPI commands at any time, select the "SCPI Recorder" icon in the toolbar.

Recording is stopped when you click "Stop Recording".

To continue recording, click "Start Recording" again.

To start a new SCPI command list, click Clear All before clicking "Start Recording".

Note:

- Some parameters cannot be set by a SCPI command.
  In this case, no SCPI command appears in the "SCPI Recorder" dialog when you record settings automatically.
- The vector network analyzer automatically clears the SCPI command list after booting.

Remote command:

`SYSTem:SRECorder[:STATe]`

List of recorded commands/script editor

The currently recorded commands are displayed in a basic editor directly in the "SCPI Recorder" dialog.

Remove Last

Deletes the last recorded SCPI command from the list.

Load Recording

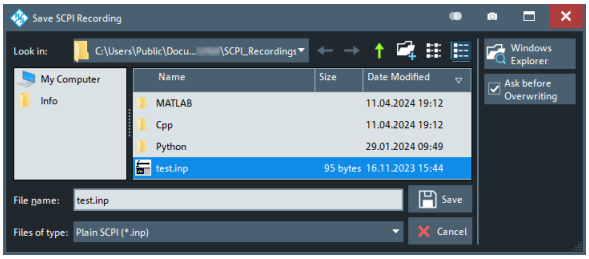
Loads an existing script in ASCII format (`*.inp`) from a file to the script editor. If the editor contains recorded commands, you must confirm a message to clear them. A file selection dialog appears.

Save Recording

Saves the current SCPI command list to the currently loaded or most recently saved `.inp` recording file.

Save As Recording

Saves the current SCPI command list to a new file in the `C:\Users\Public\Documents\Rohde-Schwarz\Vna\ScpiRecordings` directory in ASCII format with the file extension `.inp`.

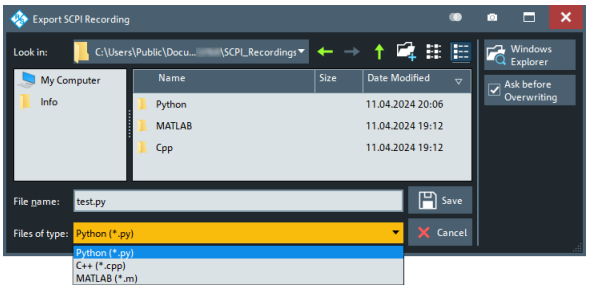


Tip: You can execute the command list in an `.inp` file using the GPIB Explorer provided with the R&S ZNB, see Section 6.1.3, "GPIB Explorer", on page 870. You can also reload `.inp` files to the script editor later.

Remote command:

`SYSTem:SRECorder:DATA?`

Export As
Exports the current SCPI command list to the specified file and directory in the selected format. By default, the file is stored in the `C:\Users\Public\Documents\Rohde-Schwarz\Vna\ScpiRecordings` directory.



Besides the recorded commands themselves, the exported script includes all format-specific header data required to execute the script using an external program on the controller. Currently, the following file formats are supported:

"Python (*.py)"	Python syntax
"C++ (*.cpp)"	C++ syntax
"MATLAB (*.m)"	MATLAB syntax

Clear All
Removes all recorded commands from the current SCPI command list.

Remote command:
`SYSTem:SRECorder:CLEar`

Help
Opens the context-sensitive help for the specific dialog.

6.6.2 How to determine the required SCPI command

The SCPI Recorder provides information on the required SCPI command for the available measurement settings, functions, and results in the "SCPI Recorder" dialog.

1. On the toolbar, click the "SCPI Recorder" icon.

   The "SCPI Recorder" dialog appears.

2. Click "Start Recording".

3. Define the setting or navigate to the function you need the SCPI command for.

   The SCPI Recorder dialog with the corresponding SCPI command appears in the "SCPI Recorder" dialog.

6.6.3 How to create and export SCPI scripts

Using the SCPI Recorder functions, you can create a SCPI script directly on the instrument and then export the script for use on the controller. The SCPI Recorder allows you to record SCPI command lists automatically.

How to record SCPI commands automatically

The following procedure explains how to record SCPI commands automatically during operation.

1. On the toolbar, click the "SCPI Recorder" icon.

   The "SCPI Recorder" dialog appears.

2. Click "Start Recording".

   The instrument starts recording all the commands required to execute the steps you perform on the instrument.

3. To stop SCPI recording, click the "SCPI Recorder" icon again.

   The "SCPI Recorder" dialog with the recorded command list appears.

4. Click "Stop Recording".

5. Save the recorded command list to a file for later use.

   a) Click "Save As Recording".
   b) Define a file name for the script file.

To edit a SCPI command list

All command lists can be edited after recording, either directly on the instrument or in any suitable editor on the controller. The following functions describe how to edit the SCPI command list directly in the "SCPI Recorder" dialog.

1. On the toolbar, click the "SCPI Recorder" icon.

   The "SCPI Recorder" dialog with the currently recorded command list appears.

2. To load a stored script in ASCII format:

   a) Click "Load Recording" in the "SCPI Recorder" dialog.
   b) Confirm the message to clear the existing commands in the editor.

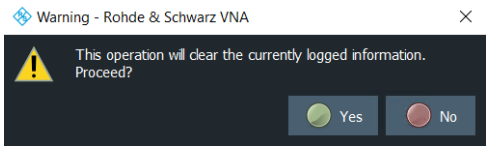


Figure 6-2: Warning message

c) Select the stored `*.inp` file.

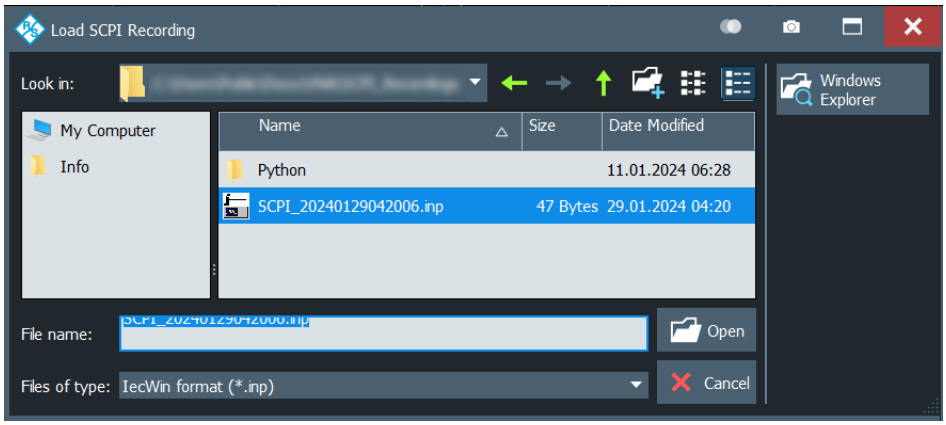


Figure 6-3: Select the recording file

d) Click "Open".

The stored commands appear in the editor.

3. To remove the most recently recorded command, click Remove Last "Remove Last" in the "SCPI Recorder" dialog.
4. To copy a command in the recorded command list:
   a) Select the command by tapping or using the mouse pointer.
   b) Right-click and select "Copy" to copy the command to the clipboard.
   Alternatively, you can select "Select All" to copy the complete command list to the clipboard.
5. Select "Save As Recording" to store the changes to the script.

How to check a SCPI script

Some suggestions on how you can check and improve a recorded SCPI script:

- Search and remove missing command entries.
  If a configured setting or performed function does not have a corresponding command, the "SCPI Recorder" dialog does not show any SCPI command in this case.
- Remove unnecessary commands written after a preset.
- Rearrange the commands to a reasonable order. For example, if you move a `STATe` command to the end of your script, you can avoid intermediate calculations of the signal.
- Check the script for completeness by comparing its results with the modified settings in manual mode.

How to export a SCPI script

When you save a command list to a file, only the recorded commands are stored in a text file. However, to execute a script in an external programming environment, it requires additional header data according to the specific format.

1. On the toolbar, click the "SCPI Recorder" icon.

The "SCPI Recorder" dialog with the currently recorded command list appears.

2. Click "Export As".
3. Define a file name and storage location for the script file.

   Note: The vector network analyzer automatically selects python as the format of the script.
4. Select "Save".

   A python script with the required header data is stored to a file.

6.6.4 How to open the context sensitive help for the SCPI commands

The SCPI Recorder allows you to view the context sensitive help for a SCPI command in a separate window.

To open the context senstive help for a SCPI command:

1. On the toolbar, click the "SCPI Recorder" icon.

   The "SCPI Recorder" dialog appears.
2. Click "Start Recording".
3. Define the setting or navigate to the function you need the SCPI command for.

   The SCPI Recorder dialog with the corresponding SCPI command appears in the "SCPI Recorder" dialog.
4. Select the command by tapping or using the mouse pointer.

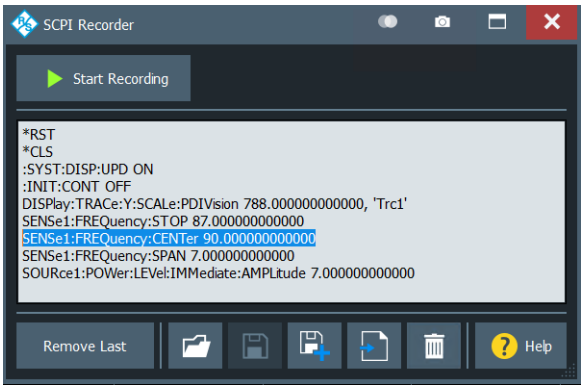


Figure 6-4: Select the SCPI command

5. Click "Help".

   The corresponding context senstive help for the SCPI command opens in a separate window.

7 Command reference

This chapter describes all common commands and SCPI commands implemented by the analyzer.

Validity of the command set

The commands reported in this chapter are valid for vector network analyzers with any number of ports. However, some of the program examples assume that more than 2 ports are available. In most cases, a simple adjustment of the port suffixes or parameters ensures compatibility with 2-port analyzers.

Compatibility with R&S ZVB and older instruments

The SCPI command set for the R&S ZNB vector network analyzer has been designed for compatibility with network analyzers R&S ZVA and R&S ZVB. A special subset of commands has been implemented for compatibility with older analyzers of the R&S ZVR family. These commands are listed in Section 7.5, "R&S ZVR/ZVABT compatible commands", on page 1620.

If you want to make full use of the R&S ZNB features but do not need R&S ZVR compatibility, you should use the commands listed in Section 7.3, "SCPI command reference", on page 917.

7.1 Special terms and notation

This section explains the meaning of special syntax elements used in the SCPI command reference sections.

The following information is provided in the reference sections:

- Complete command syntax and parameter list
- Description of the command and its relationship with other commands
- List and description of the parameters with their numerical ranges, default values and default units
- Supported command types (setting command, query). If nothing is mentioned, the command can be used to write **and** read data (setting command and query).
- Program example

The SCPI conformance information is stated at the beginning of each section. Unless otherwise stated, the commands are device-specific.

The commands are generally arranged in alphabetical order. Commands with similar function (e.g. a pair of `...STARt` and `...STOP` commands) may be described in a common section, which in some instances disrupts the strict alphabetical order.

7.1.1 Upper- vs. lower-case

Upper-/lower-case characters characterize the long and short form of the mnemonics in a command. The short form consists of all uppercase characters, the long form of all uppercase plus all lowercase characters. It is recommended to use either the short form or the long form; mixed forms are not always recognized. The R&S ZNB itself does not distinguish uppercase and lowercase characters.

7.1.2 Special characters

The following special characters are frequently used in the command description:

- **|** A vertical stroke characterizes alternative parameter settings. Only one of the parameters separated by | must be selected.
- **[ ]** Mnemonics in square brackets can be omitted when composing the command header. The complete command must be recognized by the instrument for reasons of compatibility with the SCPI standard. Parameters in square brackets are optional as well. They may be used in some application contexts, omitted in others.
- **{ }** Braces or curly brackets enclose one or more parameters that may be included zero or more times.

7.1.3 Parameters

Many commands are supplemented by a parameter or a list of parameters. Parameters either provide alternative options (setting a or setting b or setting c ..., see special character "|"), or they form a list separated by commas (setting x, y).

- **<Parameter1>, <Parameter2>...:** In the command tables and lists, parameters are generally described by a name (Parameter1, Parameter2...) written in angle brackets (<>). In an application program, <Parameter1>, <Parameter2>... must be replaced by one of the possible settings reported in the detailed parameter description.
  Example: `CONTrol:AUXiliary:C[:DATA] <DecValue>`
  with <DecValue> = 0 to 255
  --> possible command syntax: `CONTrol:AUXiliary:C:DATA 1`
- **NAN (Not A Number)** (as a returned value) is generally used to represent missing data, e.g. if a portion of a trace has not been acquired yet. It is also returned after invalid mathematical operations such as division by zero. As defined in the SCPI standard, NAN is represented as 9.91 E 37.
- **INV (INValid)** is returned, e.g., if a limit check is performed without defining the appropriate tolerance values.

7.1.4 Numeric suffixes

Symbols in angular brackets (<Ch>, <Chn>, <Mk>...) are numeric suffixes. Numeric suffixes are replaced by integer numbers to distinguish various items of the same type.

The analyzer provides numeric suffixes for channels, traces, ports, markers etc. If unspecified, a numeric suffix is replaced by 1.

The number of ports depends on the analyzer model. No restrictions apply to number of markers, channel, trace, and diagram suffixes.

In remote control, one active trace can be selected for each channel; see Section 6.3.2, "Active traces in remote control", on page 883. This concept simplifies the remote control command syntax, because it allows the active trace in a particular channel to be referenced by the channel suffix.

To keep the syntax transparent,

- <Ch> is used for channel settings (<Ch> refers to the configured channel)
- <Chn> is used for trace settings (<Chn> refers to the channel whose active trace is configured)

7.2 Common commands

Common commands are described in the IEEE 488.2 (IEC 625-2) standard. These commands have the same effect and are employed in the same way on different devices. The headers of these commands consist of "*" followed by three letters. Many common commands are related to the status reporting system; see Section 7.3.19, "TRIGger commands", on page 1589.

***CLS**

Clear status

Sets the status byte (STB), the standard event register (ESR) and the `EVENt` part of the `QUEStionable` and the `OPERation` registers to zero. The command does not alter the mask and transition parts of the registers. It clears the output buffer.

Usage: Setting only

***ESE** <Value>

Event status enable

Sets the event status enable register to the specified value. The query returns the contents of the event status enable register in decimal form.

Parameters:
<Value> Range: 0 to 255

***ESR?**

Event status read

Returns the contents of the event status register in decimal form and then sets the register to zero.

Return values:
<Contents> Range: 0 to 255

Usage: Query only

***IDN?**

Identification

Returns the instrument identification string of the R&S ZNB, including the manufacturer, the instrument type, its serial number, and the software revision..

The identification string is editable; see "Define *IDN + *OPT..." on page 843.

Usage: Query only

***IST?**

Individual status query

Returns the contents of the IST flag in decimal form. The IST flag is the status bit which is sent during a parallel poll.

Return values:
<ISTflag> 0 | 1

Usage: Query only

***OPC**

Operation complete

Sets bit 0 in the event status register when all preceding commands have been executed. This bit can be used to initiate a service request. The query writes a "1" into the output buffer when all preceding commands have been executed, which is useful for command synchronization (see Section 6.4.6, "Command sequence and command synchronization", on page 888).

***OPT?**

Option identification query

Returns a list of installed options. The response consists of arbitrary ASCII response data according to IEEE 488.2. The options are returned at fixed positions in a comma-separated string. A zero is returned for options that are not installed.

For a list of all available options and their description, refer to the specifications document.

The `OPT` information is editable; see "Define *IDN + *OPT..." on page 843.

Usage: Query only

Manual operation: See "Software Option Info" on page 829

***PCB** <Address>

Pass control back

Indicates the controller address to which remote control is returned after termination of the triggered action.

Setting parameters:
<Address> Range: 0 to 30

Usage: Setting only

***PRE** <Value>

Parallel poll register enable

Sets parallel poll enable register to the indicated value. The query returns the contents of the parallel poll enable register in decimal form.

Parameters:
<Value> Range: 0 to 255

***PSC** <Action>

Power on status clear

Determines whether the contents of the `ENABle` registers are preserved or reset when the instrument is switched on. Thus a service request can be triggered when the instrument is switched on, if the status registers `ESE` and `SRE` are suitably configured. The query reads out the contents of the "power-on-status-clear" flag.

Parameters:
<Action> 0 | 1

0
The contents of the status registers are preserved.

1
Resets the status registers.

***RST**

Reset

Sets the instrument to a defined default status. The command is equivalent to `SYSTem:PRESet[:DUMMy]`. The `*RST` value of each command is reported in the reference description. See also `SYSTem:PRESet:SCOPe`.

Usage: Setting only

***SRE** <Contents>

Service request enable

Sets the service request enable register to the value indicated. Bit 6 (MSS mask bit) remains 0. This command determines under which conditions a service request is triggered. The query `*SRE?` returns the contents of the service request enable register in decimal form. Bit 6 is always 0.

Parameters:

<Contents> Contents of the service request enable register in decimal form. Bit 6 (MSS mask bit) is always 0.

Range: 0 to 255

***STB?**

Status byte query

Reads the contents of the status byte in decimal form.

Usage: Query only

***TRG**

Trigger

Triggers all actions waiting for a trigger event. In particular, `*TRG` generates a manual trigger signal. This common command complements the Section 7.3.19, "TRIGger commands", on page 1589 of the analyzer.

Usage: Event

***TST?**

Self-test query

Performs a self test and returns its boolean result (0 = passed, 1 = failed)

Return values:

<ErrorCode> **integer > 0 (in decimal format)**
An error occurred.

0
No errors occurred.

Usage: Query only

Manual operation: See "Execute" on page 830

***WAI**

Wait to continue

Prevents servicing of the subsequent commands until all preceding commands have been executed and all signals have settled (see Command sequence and command synchronization and `*OPC`).

Usage: Event

7.3 SCPI command reference

The following sections provide detailed reference information on the instrument control commands implemented by the R&S ZNB network analyzer.

- CALCulate commands 917
- CONFigure commands 1175
- CONTrol commands 1183
- DIAGnostic commands 1208
- DISPlay commands 1210
- FORMat commands 1241
- HCOPy commands 1242
- INITiate commands 1247
- INSTrument commands 1251
- MEMory commands 1252
- MMEMory commands 1254
- OUTPut commands 1300
- PROGram commands 1305
- [SENSe:] commands 1308
- SOURce commands 1482
- STATus commands 1533
- SYSTem commands 1536
- TRACe commands 1586
- TRIGger commands 1589

7.3.1 CALCulate commands

The `CALCulate...` commands perform post-acquisition data processing. Functions in the `SENSe` subsystem are related to data acquisition, while the `CALCulate` subsystem operates on the data acquired by a `SENSe` function.

- CALCulate:CALValidate... 918
- CALCulate:DATA... 920
- CALCulate:DTIMe... 930
- CALCulate:EYE... 933
- CALCulate:FILTer[:GATE]... 964
- CALCulate:FMODel... 971
- CALCulate:LIMit... 1014
- CALCulate:MARKer... 1032
- CALCulate:MATH... 1059
- CALCulate:PARameter... 1063
- CALCulate:PULSe... 1075
- CALCulate:RIPPle... 1081
- CALCulate:STATistics... 1090
- CALCulate:TRANsform... 1099
- CALCulate:TTIMe:... 1165
- CALCulate... (other) 1167

7.3.1.1 CALCulate:CALValidate...

Commands related to Calibration validation.

CALCulate:CALValidate:AVERage[:STATe] 918
CALCulate:CALValidate:FORMat 918
CALCulate:CALValidate:MODE 918
CALCulate:CALValidate:RESPonse:LMAGnitude:LOWer 919
CALCulate:CALValidate:RESPonse:LMAGnitude:UPPer 919
CALCulate:CALValidate:RESPonse:MAGNitude:LOWer 919
CALCulate:CALValidate:RESPonse:MAGNitude:UPPer 919
CALCulate:CALValidate:RESPonse:PHASe:LOWer 919
CALCulate:CALValidate:RESPonse:PHASe:UPPer 919
CALCulate:CALValidate:RESPonse:PRESet 919
CALCulate:CALValidate:RUN 920
CALCulate:CALValidate:RUN:RESult? 920
CALCulate:CALValidate:STANdard 920

CALCulate:CALValidate:AVERage[:STATe] <Boolean>

Disables/enables averaging for cal validation.

Parameters:
<Boolean> *RST: ON (1)

Manual operation: See "Preferences" on page 633

CALCulate:CALValidate:FORMat <Format>

Selects the trace format for the limit check
(`CALCulate:CALValidate:RESPonse:...`)

Parameters:
<Format> MLOGarithmic | MLINear

MLOGarithmic
Magnitude (dB) and phase

MLINear
Magnitude (linear) and phase

Manual operation: See "Preferences" on page 633

CALCulate:CALValidate:MODE <Mode>

Selects the operating mode of the calibration validation logic.

Parameters:
<Mode> BASic | ADVanced

BASic
See "Basic mode" on page 631

ADVanced
See "Advanced mode" on page 632

Manual operation: See "Advanced Mode" on page 632

CALCulate:CALValidate:RESPonse:LMAGnitude:LOWer <Port>, <Value>
CALCulate:CALValidate:RESPonse:LMAGnitude:UPPer <Port>, <Value>

Defines the magnitude cal validation limits for the `MLINear` trace format (see `CALCulate:CALValidate:FORMat`).

Parameters:

<Port> Physical analyzer port no.

<Value> *RST: -52 (LOWer) | +52 (UPPer)
Default unit: mU

Manual operation: See "Validation Limits" on page 632

CALCulate:CALValidate:RESPonse:MAGNitude:LOWer <Port>, <Value>
CALCulate:CALValidate:RESPonse:MAGNitude:UPPer <Port>, <Value>

Defines the magnitude cal validation limits for the `MLOGarithmic` trace format (see `CALCulate:CALValidate:FORMat`).

Parameters:

<Port> Physical analyzer port no.

<Value> *RST: -0.2 (LOWer) | +0.2 (UPPer)
Default unit: dB

Manual operation: See "Validation Limits" on page 632

CALCulate:CALValidate:RESPonse:PHASe:LOWer <Port>, <Value>
CALCulate:CALValidate:RESPonse:PHASe:UPPer <Port>, <Value>

Defines the phase deviation limits for calibration validation.

Note that phase limits are only checked for high reflection standards Open and Short, and not for Match (see `CALCulate:CALValidate:STANdard`).

Parameters:

<Port> Physical analyzer port number

<Value> *RST: -8 (LOWer) | +8 (UPPer)
Default unit: degrees

Manual operation: See "Validation Limits" on page 632

CALCulate:CALValidate:RESPonse:PRESet

Resets the cal validation limits (`CALCulate:CALValidate:RESPonse:<MAGNitude|LMAGnitude|PHASe>:<LOWer|UPPer>`).

Usage: Event

Manual operation: See "Validation Limits" on page 632

CALCulate:CALValidate:RUN

Runs the cal validation with its configured settings.

Usage: Event

Manual operation: See "Validate" on page 631

CALCulate:CALValidate:RUN:RESult?

Displays the result (PASS/FAIL) of the preceding cal validation (`CALCulate:CALValidate:RUN`).

Usage: Query only

Manual operation: See "Validate" on page 631

CALCulate:CALValidate:STANdard <Standard>

Selects the CalU standard to be used for cal validation in advanced mode.

Note that phase limits `CALCulate:CALValidate:RESPonse:PHASe:<LOWer|UPPer>`) are only checked for high reflection standards OPEN and SHORt, and not for MATCh.

Parameters:
<Standard> OPEN | SHORt | MATCh

Manual operation: See "Calibration Standard" on page 632

7.3.1.2 CALCulate:DATA...

The `CALCulate:DATA...` commands provide access to the results of a measurement.

Data format
The trace data is transferred in either ASCII or block data (REAL) format, depending on the `FORMat[:DATA]` setting. If block data format is used, it is recommended to select EOI as a receive terminator (`SYSTem:COMMunicate:GPIB[:SELF]:RTERminator EOI`).

CALCulate<Chn>:DATA....921
CALCulate:DATA:ALL?....924
CALCulate<Ch>:DATA:CALL....924
CALCulate<Ch>:DATA:CALL:CATalog?....925
CALCulate<Ch>:DATA:CHANnel:ALL?....925
CALCulate<Ch>:DATA:CHANnel:DALL?....926
CALCulate:DATA:DALL?....926

CALCulate<Ch>:DATA:MDATa:INTerpolate....926
CALCulate<Chn>:DATA:NSWeep:COUNt?....927
CALCulate<Chn>:DATA:NSWeep:FIRSt?....927
CALCulate<Chn>:DATA:NSWeep[:LAST]?....928
CALCulate<Ch>:DATA:SGRoup?....929
CALCulate<Chn>:DATA:STIMulus?....929
CALCulate:DATA:TRACe....929

CALCulate<Chn>:DATA <Format>, <Data>...
CALCulate<Chn>:DATA? <Format>

The query reads the response values of the selected channel's active trace or reads error terms of the selected channel.

The "set command" either imports formatted or unformatted trace data to the selected channel's active trace (see Section 4.1.5, "Data flow", on page 89) or writes error terms of the selected channel.

Note

- The data format is parameter-dependent; see tables below. The unit is the default unit of the measured parameter; see `CALCulate<Ch>:PARameter:SDEFine`.
- Unformatted trace data (SDATa) can only be imported to memory traces
- Formatted trace data (FDATa) can only be imported to "live" traces if the related channel is in single sweep mode (`INITiate<Ch>:CONTinuous OFF`).
  Before the import, the target trace must be prepared according to the settings used during export. Any request for new data from the hardware ("Restart Sweep" in single sweep mode or switching to continuous sweep mode) discards imported data and switches the display back to measured data.

Suffix:

<Chn> Channel number used to identify the active trace

Parameters:

<Data> Trace data either in ASCII or block data format, depending on the current `FORMat[:DATA]` setting.

Parameters for setting and query:

<Format> FDATa | SDATa | MDATa | NCData | UCData | SCORr1 | SCORr2 | SCORr3 | SCORr4 | SCORr5 | SCORr6 | SCORr7 | SCORr8 | SCORr9 | SCORr10 | SCORr11 | SCORr12 | SCORr13 | SCORr14 | SCORr15 | SCORr16 | SCORr17 | SCORr18 | SCORr19 | SCORr20 | SCORr21 | SCORr22 | SCORr23 | SCORr24 | SCORr25 | SCORr26 | SCORr27

See list of parameters below.

Example:

```
*RST; SWE:POIN 20
```
Create a trace with 20 sweep points, making the created trace the active trace of channel 1 (omitted optional mnemonic SENSe1).
```
CALC:DATA? FDAT
```
Query the 20 response values of the created trace. In the FDATa setting, 20 comma-separated ASCII values are returned.
```
CALC:DATA:STIM?
```
Query the 20 stimulus values of the created trace. 20 comma-separated ASCII values are returned.
```
CALC2:PAR:SDEF 'Trc2', 'S11'
```
Create a second trace in a new channel no. 2. The second trace does not become the active trace and is not displayed.
```
CALC:DATA:TRAC? 'Trc2', FDAT
```
Query the response values of the second (non-active) trace. 20 comma-separated ASCII values are returned.
```
CALC:DATA:ALL? FDAT
```
Query the response values of all traces. 40 comma-separated ASCII values are returned.

Example: **Writing memory traces**
```
*RST; SWE:POIN 3
```
Create a data trace 'Trc1' with 3 sweep points, making the created trace the active trace of channel 1 (omitted optional mnemonic SENSe1).
```
TRAC:COPY 'MemTrc1','Trc1'; :CALC:PAR:SEL 'MemTrc1'
```
Copy the data trace to a memory trace and select the memory trace as an active trace.
```
CALC:DATA SDAT, 1,2, 3,4, 5,6
```
Write numbers (1,2), (3,4), (5,6) to the memory trace.
```
CALC:DATA? SDAT
```
Query the memory trace. The response is 1,2,3,4,5,6.
```
FORM REAL,32
```
Change the data format to 4-byte block data.
```
CALC:DATA SDAT, #2241234567890123456789012345678901234
```
Write 24 bytes (= 4 * 2 * 3 bytes) of data to the memory trace.

The following parameters are related to trace data (see also diagram in Section 4.1.5, "Data flow", on page 89):

Table 7-1: Data format identifiers used in the CALCulate:DATA... commands

FDATa	Formatted trace data, according to the selected trace format (`CALCulate<Chn>:FORMat`). One value per trace point for Cartesian diagrams, two values for polar diagrams. [Data access point 6]
SDATa	Unformatted trace data: real and imaginary part of each measurement point. Two values per trace point irrespective of the selected trace format. The trace mathematics is ignored. [Data access point 4]

MDATa	Unformatted trace data (see SDATa) after evaluation of trace mathematics. [Data access point 5]
NCData	Factory calibrated trace data: the values are obtained right after applying the factory calibration but before applying a user-defined calibration (if any). [Data access point 1]
UCData	Uncalibrated trace data. [Data access point 0] Note: • The respective trace must represent a wave quantity or ratio • The driving port and receiving port must not be on a switch matrix Otherwise an error occurs.

The following parameters denote the error terms generated during a calibration.

Table 7-2: Error terms in the CALCulate:DATA... commands

Error Term	Description	Receive Ports (S-parameter)
SCORr1, ..., SCORr12	2-port error terms; see `[SENSe<Ch>:]CORRection:DATA`.	1 and 2 (S11, S12, S21, S22)
SCORr13	Directivity	3 (S33)
SCORr14	Source match	3 (S33)
SCORr15	Reflection tracking	3 (S33)
SCORr16	Isolation	3 (S31)
SCORr17	Load match	3 (S31)
SCORr18	Transmission tracking	3 (S13)
SCORr19	Isolation	1 (S13)
SCORr20	Load match	1 (S13)
SCORr21	Transmission tracking	1 (S13)
SCORr22	Isolation	3 (S32)
SCORr23	Load match	3 (S32)
SCORr24	Transmission tracking	3 (S32)
SCORr25	Isolation	2 (S23)
SCORr26	Load match	2 (S23)
SCORr27	Transmission tracking	2 (S23)

Note: The error terms are channel-specific; they apply to the active calibration of channel no. `<Chn>` or to the factory calibration (if no channel calibration is active). For the factory calibration, the query form is allowed only (no change of factory calibration data).

Tip: Use the generalized command `[SENSe<Ch>:]CORRection:CDATa` to read or write error terms for arbitrary analyzer ports. For additional programming examples, refer to Section 8.2.5.4, "Saving and recalling error terms", on page 1676.

CALCulate:DATA:ALL? <Format>[, <RecallSet>]

Reads the current response values of all traces of the referenced recall set.

Query parameters:

<Format> FDATa | SDATa | MDATa
Output format for the S-parameter data, see CALCulate<Chn>:DATA.

<RecallSet> Recall set name; if omitted the active recall set is used

Return values:

<Data> Response values either in ASCII or block data format, depending on the current FORMat[:DATA] setting.

Example: See CALCulate<Chn>:DATA

Usage: Query only

CALCulate<Ch>:DATA:CALL <Format>, <Data>...

The query reads the current response values of all S-parameter data traces at channel <Ch>'s data access point <DACPoint>.

If a full n-port system error correction is active in the related channel, the command reads the full nxn S-matrix of the calibrated ports (there is no need to create or display the S-parameter traces). Use CALCulate<Ch>:DATA:CALL:CATalog? to query the available traces.

The "set direction" allows you to import measurement data at the "Fixture Simulation Input" data access point (see Section 4.1.5, "Data flow", on page 89).

Note

- Importing data is only supported in single sweep mode (INITiate<Ch>:CONTinuous OFF)
- Before importing data, the channel must be configured with the same settings that were used during export (user calibration, balanced port configuration, stimulus axis etc.).
  Furthermore the switch matrix configuration must match.
- After importing data, "downstream" parameters in the data flow can be changed and their effect is shown directly
- Any request for new data from the hardware (Restart Sweep in single sweep mode or switching to continuous sweep mode) discards imported data and switches the display back to measured data

Suffix:

<Ch> Channel number

Parameters:

<Format> SDATa | FSIData

SDATa
Output as unformatted trace data; see `CALCulate<Chn>:DATA`. Query only.

FSIData
Output or input at "Fixture Simulation Input" data access point.

<Data> Trace data either in ASCII or block data format, depending on the current `FORMat[:DATA]` setting.
The column order must match the one used during export.

Example: Suppose that a TOSM calibration for ports 1 and 2 is active in channel no. 1.
`CALCulate:DATA:CALL:CATalog?`
Query the traces available for `CALCulate<Ch>:DATA:CALL?`.
The response is `'S11,S12,S21,S22'`.
`CALCulate:DATA:CALL? SDATa`
Return the complex response values of all traces. The traces in the catalog list are read one after another: The response array contains n (number of points) pairs of real and imaginary values for S_{11}, followed by n pairs of values for S_{12}, S_{21}, and S_{22}.

CALCulate<Ch>:DATA:CALL:CATalog?

Returns all traces which are available for `CALCulate<Ch>:DATA:CALL` in channel no. `<Ch>`. The response is a string parameter with all S-parameter traces in the current channel or in the active system error correction; see example.

Suffix:
<Ch> Channel number

Example: See `CALCulate<Ch>:DATA:CALL`

Usage: Query only

CALCulate<Ch>:DATA:CHANnel:ALL? <Format>

Reads the current response values of all traces of the selected channel.

Suffix:
<Ch> Channel number

Query parameters:
<Format> FDATa | SDATa | MDATa
Output format for the S-parameter data, see `CALCulate<Chn>:DATA`.

Return values:
<Data> Trace data either in ASCII or block data format, depending on the current `FORMat[:DATA]` setting.

Usage: Query only

CALCulate<Ch>:DATA:CHANnel:DALL? <Format>

Reads the current response values of all data traces of the selected channel. Use `CALCulate<Ch>:DATA:CHANnel:ALL?` to query data traces and memory traces.

Suffix:
<Ch> Channel number

Query parameters:
<Format> FDATa | SDATa | MDATa
Output format for the S-parameter data, see `CALCulate<Chn>:DATA`.

Return values:
<Data> Trace data either in ASCII or block data format, depending on the current `FORMat[:DATA]` setting.

Usage: Query only

CALCulate:DATA:DALL? <Format>

Reads the current response values of all data traces of the current recall set. Use `CALCulate:DATA:ALL?` to query data traces and memory traces.

Query parameters:
<Format> FDATa | SDATa | MDATa
Output format for the S-parameter data, see `CALCulate<Chn>:DATA`.

Return values:
<Data> Response values either in ASCII or block data format, depending on the current `FORMat[:DATA]` setting.

Example: Analogous to `CALCulate:DATA:DALL?`; see `CALCulate<Chn>:DATA`.

Usage: Query only

CALCulate<Ch>:DATA:MDATa:INTerpolate

Uses linear inter-/extrapolation to "regrid" all memory traces of the related channel to the channel's current stimulus values.

Suffix:
<Ch> Channel number

Usage: Event

CALCulate<Chn>:DATA:NSWeep:COUNt?

Reads the number of completed sweeps in single sweep mode (`INITiate<Ch>:CONTinuous OFF`). The trace can be any of the traces acquired during the single sweep cycle.

Tip:

This command can only be used for `[SENSe<Ch>:]SWEep:COUNt` > 1.

Suffix:

<Chn> Channel number used to identify the active trace

Example: See `CALCulate<Chn>:DATA:NSWeep:FIRSt?`

Usage: Query only

CALCulate<Chn>:DATA:NSWeep:FIRSt? <Format>, <FwCount>[, <FwCountEnd>]

Reads the response values of a trace or a consecutive group of traces acquired in single sweep mode (`INITiate<Ch>:CONTinuous OFF`).

Tip:

This command can only be used for `[SENSe<Ch>:]SWEep:COUNt` > 1.

Suffix:

<Chn> Channel number used to identify the active trace

Query parameters:

<Format> SDATa

Read unformatted sweep data (fixed parameter): Returns the real and imaginary part of each measurement point (2 values per trace point irrespective of the selected trace format).

<FwCount> Number of the first sweep to be read. 1 denotes the first sweep acquired, 2 denotes the second, and so forth. The sweep count in single sweep mode is defined via `[SENSe<Ch>:]SWEep:COUNt`.

Range: 1 to sweep count

<FwCountEnd> Number of the last sweep to be read. If this parameter is omitted, it is implicitly set to `<FwCount>` (a single sweep is read).

Range: <FwCount> to sweep count

Return values:

<Data> Response values, either in ASCII or block data format, depending on the current `FORMat[:DATA]` setting.

Example: `SWE:COUN 10`
Define the number of sweeps (10) to be measured in single sweep mode.
`INIT:CONT OFF; :INIT;`
Activate single sweep mode and start a single sweep sequence in channel no. 1. No synchronization is necessary.
`if (CALC:DATA:NSW:COUN? > 4)`
`CALC:DATA:NSW:FIRS? SDAT, 5`
Wait until 5 sweeps have been measured, then query the results of the 5th sweep.
See also Section 8.2.4.3, "Retrieving the results of previous sweeps", on page 1670.

Usage: Query only

CALCulate<Chn>:DATA:NSWeep[:LAST]? <Format>, <RvCount>

Reads the response values of a trace acquired in single sweep mode (`INITiate<Ch>:CONTinuous OFF`). The trace can be any of the traces acquired during the single sweep cycle.

Tip:

- This command can only be used for `[SENSe<Ch>:]SWEep:COUNt` > 1.
- Ensure that the single sweep is terminated before using this command, otherwise the results of the trace count can be unpredictable (see example below). Alternatively, use the `CALCulate<Chn>:DATA:NSWeep:FIRSt?` command.

Suffix:

<Chn> Channel number used to identify the active trace

Query parameters:

<Format> SDATa

Read unformatted sweep data (fixed parameter): Returns the real and imaginary part of each measurement point (2 values per trace point, irrespective of the selected trace format).

<RvCount> Number of sweeps to be read. 1 denotes the last sweep acquired, 2 denotes the second-last, and so forth.

Range: 1 to sweep count defined via [SENSe<Ch>:]SWEep:COUNt

Example: `SWE:COUN 10`
Define the number of sweeps (10) to be measured in single sweep mode.
`INIT:CONT OFF; :INIT; *OPC?`
Activate the single sweep mode and start a single sweep sequence in channel no. 1. Wait until the single sweep sequence is complete.
`CALC:DATA:NSW? SDAT,3`
Query the results of the 8th sweep.
See also Section 8.2.4.3, "Retrieving the results of previous sweeps", on page 1670.

Usage: Query only

CALCulate<Ch>:DATA:SGRoup? <Format>

Reads the current response values of all S-parameters associated to a group of logical ports (S-parameter group). The S-parameter group must be created before using `CALCulate<Ch>:PARameter:DEFine:SGRoup`.

Suffix:
<Ch> Channel number of the previously defined S-parameter group

Query parameters:
<Format> FDATa | SDATa | MDATa
Output format for the S-parameter data, see `CALCulate<Chn>:DATA` on page 921.

Example: See `CALCulate<Ch>:PARameter:DEFine:SGRoup`

Usage: Query only

CALCulate<Chn>:DATA:STIMulus?

Reads the stimulus values of the active data or memory trace.

Suffix:
<Chn> Channel number used to identify the active trace

Example: See `CALCulate<Chn>:DATA`

Usage: Query only

CALCulate:DATA:TRACe <TraceName>, <Format>, <Data>...

The query gets the trace data of an arbitrary (not necessarily the active) trace, referenced by its trace name <TraceName>.

The "set direction" allows to import formatted or unformatted trace data to an existing trace (data access point 3 or 2 in Section 4.1.5, "Data flow", on page 89).

Note

- Unformatted trace data (SDATa) can only be imported to memory traces

- Formatted trace data (FDATa) can only be imported to "live" traces if the related channel is in single sweep mode (`INITiate<Ch>:CONTinuous OFF`).
  Before the import, the target trace must be prepared according to the settings used during export. Any request for new data from the hardware ("Restart Sweep" in single sweep mode or switching to continuous sweep mode) discards imported data and switches back the display to measured data.

Parameters:

<TraceName> String parameter containing the trace name

<Format> FDATa | SDATa | MDATa | NCData | UCData
Data format; see Table 7-1.

<Data> Trace data for `FDATa | SDATa` import either in ASCII or block data format, depending on the current `FORMat[:DATA]` setting. The column order must match the one used during export.

Example: See `CALCulate<Chn>:DATA`.

7.3.1.3 CALCulate:DTIMe...

Defines the properties and retrieves the results of the skew measurement provided with the Extended time domain analysis option R&S ZNB-K20.

CALCulate<Chn>:DTIMe:DATA? 930
CALCulate<Chn>:DTIMe:LIMit:FAIL? 931
CALCulate<Chn>:DTIMe:LIMit:FAIL:BEEP 931
CALCulate<Chn>:DTIMe:LIMit:LIMit 932
CALCulate<Chn>:DTIMe:LIMit:STATe 932
CALCulate<Chn>:DTIMe:POSition 932
CALCulate<Chn>:DTIMe:STATe 932
CALCulate<Chn>:DTIMe:TARGet 933

CALCulate<Chn>:DTIMe:DATA? [<Data>]

The setting shows or hides the skew measurement's "Extended Info".

The query returns the results of the skew measurement.

Suffix:

<Chn> Channel number used to identify the active trace

Query parameters:

<Data> ALL

Setting:
`CALCulate<Chn>:DTIMe:DATA ALL` turns the "Extended Info" on, `CALCulate<Chn>:DTIMe:DATA` turns it off.

Query:

If omitted, a single numeric value is returned. If ALL is specified, the result consists of 6 numeric values. Furthermore, the interpretation of the result values depends on the active trace's stimulus axis (`CALCulate<Chn>:TRANsform:TIME:XAXis`). See the table below.

Usage: Query only

Options: R&S ZNB-K20

Manual operation: See "Extended Info" on page 404

	<Data> omitted	**<Data> = ALL**
Stimulus axis: time	<skew time>	<skew time>,<skew distance>, <X value of ref. point on current trace>, <Y value of ref. point on current trace>, <X value of ref. point on reference trace>, <Y value of ref. point on reference trace>
Stimulus axis: distance	<skew distance>	<skew distance>, <skew time>, <X value of ref. point on current trace>, <Y value of ref. point on current trace>, <X value of ref. point on reference trace>, <Y value of ref. point on reference trace>

Note that the skew value is positive, if, at the defined position (`CALCulate<Chn>:DTIMe:POSition`), the active trace is to the right of the reference trace.

CALCulate<Chn>:DTIMe:LIMit:FAIL? <Boolean>

Indicates whether the skew has passed or failed.

Suffix:
<Chn> Channel number used to identify the active trace

Return values:
<Boolean> 0 – skew check has passed
1 – skew check has failed

Usage: Query only

Options: R&S ZNB-K20

CALCulate<Chn>:DTIMe:LIMit:FAIL:BEEP <Boolean>

Deprecated command: Since version 3.45, generating sounds is no longer supported by the analyzer firmware.

Defines whether the R&S ZNB makes an audible beep on skew limit violations.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<Boolean> *RST: OFF

Options: R&S ZNB-K20

CALCulate<Chn>:DTIMe:LIMit:LIMit <arg0>

Defines the limit value for the skew check.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<arg0> Limit value in seconds or meters, depending on the format of the current trace's stimulus axis (see `CALCulate<Chn>:TRANsform:TIME:XAXis`).

Options: R&S ZNB-K20

Manual operation: See "Skew Limit" on page 404

CALCulate<Chn>:DTIMe:LIMit:STATe <Boolean>

Activates/deactivates the skew limit check.

Use `CALCulate<Chn>:DTIMe:LIMit:LIMit` to set the applicable limit.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<Boolean> *RST: OFF

Options: R&S ZNB-K20

Manual operation: See "Skew Check On" on page 404

CALCulate<Chn>:DTIMe:POSition <SkewPos>

Defines the position of the skew measurement.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<SkewPos> Skew position as integer percentage of the step size

Options: R&S ZNB-K20

Manual operation: See "Skew Position" on page 404

CALCulate<Chn>:DTIMe:STATe <Boolean>

Enables/disables the Skew measurement.

Note that the skew measurement can only be performed if the following conditions are met for both the active trace and the reference trace:

- The trace format is real (`CALCulate<Chn>:FORMat REAL`)
- Time Domain is enabled (`CALCulate<Chn>:TRANsform:TIME:STATe ON`)

- The Low Pass Step time domain transform is used `CALCulate<Chn>:TRANsform:TIME[:TYPE] LPASs` and `CALCulate<Chn>:TRANsform:TIME:STIMulus STEP`)

The latter, in turn, requires the stimulus grid to be harmonic. A harmonic grid can be achieved, for example, using `[SENSe<Ch>:]HARMonic:AUTO ON`.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<Boolean> *RST: OFF

Options: R&S ZNB-K20

Manual operation: See "Skew" on page 403

CALCulate<Chn>:DTIMe:TARGet <SkewRefTrace>

Selects the reference trace for the skew measurement.

See `CALCulate<Chn>:DTIMe:STATe` for conditions on both the active and the reference trace.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<SkewRefTrace> Reference trace, identified by its name (within channel <Chn>).

Options: R&S ZNB-K20

Manual operation: See "Reference Trace" on page 404

7.3.1.4 CALCulate:EYE...

Defines the properties and retrieves the results of the eye diagram measurement provided with the Extended time domain analysis option R&S ZNB-K20.

CALCulate<Chn>:EYE:DUT:MODE....935
CALCulate<Chn>:EYE:EMPHasis:CURSor:POST<1|2>....935
CALCulate<Chn>:EYE:EMPHasis:CURSor:PRE....936
CALCulate<Chn>:EYE:EMPHasis:STATe....936
CALCulate<Chn>:EYE:EQUalization:CTLE:COUNt:POLE....936
CALCulate<Chn>:EYE:EQUalization:CTLE:COUNt:ZERO....937
CALCulate<Chn>:EYE:EQUalization:CTLE:DC....937
CALCulate<Chn>:EYE:EQUalization:CTLE:LIST:POLE....937
CALCulate<Chn>:EYE:EQUalization:CTLE:LIST:ZERO....938
CALCulate<Chn>:EYE:EQUalization:CTLE:POLE<Num>....938
CALCulate<Chn>:EYE:EQUalization:CTLE:RATio....938
CALCulate<Chn>:EYE:EQUalization:CTLE:STATe....939
CALCulate<Chn>:EYE:EQUalization:CTLE:ZERO<Num>....939
CALCulate<Chn>:EYE:EQUalization:DFE:COUNt:TAPS....939
CALCulate<Chn>:EYE:EQUalization:DFE:LIST:TAPS....940

CALCulate<Chn>:EYE:EQUalization:DFE:MODE....940
CALCulate<Chn>:EYE:EQUalization:DFE:STATe....940
CALCulate<Chn>:EYE:EQUalization:DFE:TAP<Num>....940
CALCulate<Chn>:EYE:EQUalization:FFE:COUNt:POSTcursor....941
CALCulate<Chn>:EYE:EQUalization:FFE:COUNt:PRECursor....941
CALCulate<Chn>:EYE:EQUalization:FFE:LIST:POSTcursor....941
CALCulate<Chn>:EYE:EQUalization:FFE:LIST:PRECursor....941
CALCulate<Chn>:EYE:EQUalization:FFE:MAIN:TAP....942
CALCulate<Chn>:EYE:EQUalization:FFE:MODE....942
CALCulate<Chn>:EYE:EQUalization:FFE:POSTcursor<Num>....942
CALCulate<Chn>:EYE:EQUalization:FFE:PRECursor<Num>....943
CALCulate<Chn>:EYE:EQUalization:FFE:STATe....943
CALCulate<Chn>:EYE:EQUalization:STATe....943
CALCulate<Chn>:EYE:INPut:BPATtern:TYPE....944
CALCulate<Chn>:EYE:INPut:DRATe....944
CALCulate<Chn>:EYE:INPut:LENGth:BITS....944
CALCulate<Chn>:EYE:INPut:LENGth:PRBS....945
CALCulate<Chn>:EYE:INPut:MODulation....945
CALCulate<Chn>:EYE:INPut:XRATe....946
CALCulate<Chn>:EYE:INPut:OLEVel....946
CALCulate<Chn>:EYE:INPut:ZLEVel....946
CALCulate<Chn>:EYE:INPut:RTIMe:DATA....946
CALCulate<Chn>:EYE:INPut:RTIMe:THReshold....947
CALCulate<Chn>:EYE:JITTer:DIRac:DELTa....947
CALCulate<Chn>:EYE:JITTer:DIRac:PROBability....947
CALCulate<Chn>:EYE:JITTer:PERiodic:FREQuency....948
CALCulate<Chn>:EYE:JITTer:PERiodic:MAGNitude....948
CALCulate<Chn>:EYE:JITTer:PERiodic:PHASe....949
CALCulate<Chn>:EYE:JITTer:RANDom:STDDeviation....949
CALCulate<Chn>:EYE:JITTer:STATe....949
CALCulate<Chn>:EYE:JITTer:TYPE:DIRac....950
CALCulate<Chn>:EYE:JITTer:TYPE:PERiodic....950
CALCulate<Chn>:EYE:JITTer:TYPE:RANDom....951
CALCulate<Chn>:EYE:JITTer:TYPE:USER....951
CALCulate<Chn>:EYE:MASK AUTO....952
CALCulate<Chn>:EYE:MASK:CENTer:HORizontal....952
CALCulate<Chn>:EYE:MASK:CENTer:VERTical....952
CALCulate<Chn>:EYE:MASK:DATA?....952
CALCulate<Chn>:EYE:MASK:FAIL?....953
CALCulate<Chn>:EYE:MASK:FAIL:BEEP....954
CALCulate<Chn>:EYE:MASK:FAIL:CONDition....954
CALCulate<Chn>:EYE:MASK:SHAPe:BOTTom:HORizontal....955
CALCulate<Chn>:EYE:MASK:SHAPe:TOP:HORizontal....955
CALCulate<Chn>:EYE:MASK:SHAPe:BOTTom:STATe....955
CALCulate<Chn>:EYE:MASK:SHAPe:POLYgon:STATe....955
CALCulate<Chn>:EYE:MASK:SHAPe:TOP:STATe....955
CALCulate<Chn>:EYE:MASK:SHAPe:BOTTom:VERTical....955
CALCulate<Chn>:EYE:MASK:SHAPe:TOP:VERTical....955
CALCulate<Chn>:EYE:MEASurement:BASic....956
CALCulate<Chn>:EYE:MEASurement:TIME....956

CALCulate<Chn>:EYE:MASK:SHAPe:POLYgon:HORizontal....956
CALCulate<Chn>:EYE:MASK:SHAPe:POLYgon:TYPE....957
CALCulate<Chn>:EYE:MASK:SHAPe:POLYgon:VERTical....957
CALCulate<Chn>:EYE:MASK:SHOW....958
CALCulate<Chn>:EYE:MASK:STATe....958
CALCulate<Chn>:EYE:MASK:VIOLation:RATE....958
CALCulate<Chn>:EYE:MASK:VIOLation:TOLerance....959
CALCulate<Chn>:EYE:MEASurement:DATA?....959
CALCulate<Chn>:EYE:MEASurement:STATe....960
CALCulate<Chn>:EYE:MEASurement:TTIMe:THReshold....961
CALCulate<Chn>:EYE:NOISe:RMS....961
CALCulate<Chn>:EYE:NOISe:STATe....961
CALCulate<Chn>:EYE:STATe....962
CALCulate<Chn>:EYE:STIMulus:ENCoder....962
CALCulate<Chn>:EYE:STIMulus:LOWPass....962
CALCulate<Chn>:EYE:STIMulus:SCRambler....963
CALCulate<Chn>:EYE:VIEW....963

CALCulate<Chn>:EYE:DUT:MODE <DUTMode>

Allows to (temporarily) switch between the measured DUT and an ideal one (with flat frequency response) in the related eye diagram.

This command will raise an execution error if the active trace in the selected channel is not an eye diagram.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<DUTMode> IDEal | MEASured

Options: R&S ZNB-K20

Manual operation: See "Mode" on page 394

CALCulate<Chn>:EYE:EMPHasis:CURSor:POST<1|2> <Weight>

Sets the weights of the post-cursor taps for the pre-emphasis FIR filter in the calculation chain of the related eye diagram.

This command will raise an execution error if the active trace in the selected channel is not an eye diagram.

Emphasis in the generator simulation for eye diagram measurements is enabled using `CALCulate<Chn>:EYE:EMPHasis:STATe`.

Suffix:
<Chn> Channel number used to identify the active trace

<1|2> 1 for the "Post 1" filter tap, 2 for the "Post 2" filter tap.

Parameters:
<Weight> Weight relative to the "Cursor" tap
*RST: 0 dB
Default unit: dB

Options: R&S ZNB-K20

Manual operation: See "Cursor Settings" on page 390

CALCulate<Chn>:EYE:EMPHasis:CURSor:PRE <Weight>

Sets the weight of the pre-cursor tap for the pre-emphasis FIR filter in the calculation chain of the related eye diagram.

This command will raise an execution error if the active trace in the selected channel is not an eye diagram.

Emphasis in the generator simulation for eye diagram measurements is enabled using `CALCulate<Chn>:EYE:EMPHasis:STATe`.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<Weight> Weight relative to the "Cursor" tap
*RST: 0 dB
Default unit: dB

Options: R&S ZNB-K20

Manual operation: See "Cursor Settings" on page 390

CALCulate<Chn>:EYE:EMPHasis:STATe <State>

Activates/deactivates pre-emphasis in the calculation chain of the related eye diagram.

This command will raise an execution error if the active trace in the selected channel is not an eye diagram.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<State> *RST: OFF

Options: R&S ZNB-K20

Manual operation: See "Active" on page 390

CALCulate<Chn>:EYE:EQUalization:CTLE:COUNt:POLE <Count>

Defines the number of poles in the equalizer function.

Suffix:
<Chn> The channel number used to identify the active trace.

Parameters:
<Count> Default: 2

Options: R&S ZNB-K20

Manual operation: See "Poles, Frequency" on page 395

CALCulate<Chn>:EYE:EQUalization:CTLE:COUNt:ZERO <Count>

Defines the number of zero transitions in the equalizer function.

Suffix:
<Chn> The channel number used to identify the active trace.

Parameters:
<Count> Default: 1

Options: R&S ZNB-K20

Manual operation: See "Zeros, Frequency" on page 395

CALCulate<Chn>:EYE:EQUalization:CTLE:DC <DcGain>

Specifies the DC gain of the CTLE (a two-pole filter with single zero) at the receiver simulation of the related eye diagram.

This command will raise an execution error if the active trace in the selected channel is not an eye diagram.

The equalizer can be enabled/disabled using `CALCulate<Chn>:EYE:EQUalization:STATe`.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<DcGain> *RST: 0 dB
Default unit: dB

Options: R&S ZNB-K20

Manual operation: See "DC Gain, Ratio" on page 394

CALCulate<Chn>:EYE:EQUalization:CTLE:LIST:POLE <Poles>...

Lists the frequencies of the poles of the CTLE function.

The number of expected frequencies is defined by `CALCulate<Chn>:EYE:EQUalization:CTLE:COUNt:POLE` on page 936.

Suffix:
<Chn> The channel number used to identify the active trace.

Parameters:
<Poles> Numeric value list
Default unit: Hz

Options: R&S ZNB-K20

Manual operation: See "Poles, Frequency" on page 395

CALCulate<Chn>:EYE:EQUalization:CTLE:LIST:ZERO <Zeros>...

Lists the frequencies of the zero transitions of the CTLE function.

The number of expected frequencies is defined by `CALCulate<Chn>:EYE:EQUalization:CTLE:COUNt:ZERO` on page 937.

Suffix:
<Chn> The channel number used to identify the active trace.

Parameters:
<Zeros> Numeric value list
Default unit: Hz

Options: R&S ZNB-K20

Manual operation: See "Zeros, Frequency" on page 395

CALCulate<Chn>:EYE:EQUalization:CTLE:POLE<Num> <Freq>

Specifies the poles of the CTLE (a two-pole filter with single zero) used at the receiver simulation of the related eye diagram.

This command will raise an execution error if the active trace in the selected channel is not an eye diagram.

The equalizer can be enabled/disabled using `CALCulate<Chn>:EYE:EQUalization:STATe`.

Suffix:
<Chn> Channel number used to identify the active trace

<Num>

Parameters:
<Freq> Default unit: Hz

Options: R&S ZNB-K20

Manual operation: See "Poles, Frequency" on page 395

CALCulate<Chn>:EYE:EQUalization:CTLE:RATio <Ratio>

Ratio of the output to input voltage.

Suffix:
<Chn> The channel number used to identify the active trace.

Parameters:
<Ratio> Numeric value

Options: R&S ZNB-K20

Manual operation: See "DC Gain, Ratio" on page 394

CALCulate<Chn>:EYE:EQUalization:CTLE:STATe <State>

Enables or disables CTLE equalization for the eye diagram.

Suffix:
<Chn> The channel number used to identify the active trace.

Parameters:
<State> ON | OFF - enables or disables equalization.

Options: R&S ZNB-K20

Manual operation: See "CTLE" on page 394

CALCulate<Chn>:EYE:EQUalization:CTLE:ZERO<Num> <Freq>

Specifies the zero of the equalizer (a two-pole filter with single zero) used at the receiver simulation of the related eye diagram.

This command will raise an execution error if the active trace in the selected channel is not an eye diagram.

The equalizer can be enabled/disabled using `CALCulate<Chn>:EYE:EQUalization:STATe`.

Suffix:
<Chn> Channel number used to identify the active trace

<Num>

Parameters:
<Freq> Default unit: Hz

Options: R&S ZNB-K20

Manual operation: See "Zeros, Frequency" on page 395

CALCulate<Chn>:EYE:EQUalization:DFE:COUNt:TAPS <Count>

Defines the number of taps for DFE equalization.

Suffix:
<Chn> The channel number used to identify the active trace.

Parameters:
<Count> Numeric value

Options: R&S ZNB-K20

Manual operation: See "Taps, Weight" on page 396

CALCulate<Chn>:EYE:EQUalization:DFE:LIST:TAPS <Taps>...

Lists the weights for the DFE taps.

Suffix:
<Chn> The channel number used to identify the active trace.

Parameters:
<Taps> List of numeric values between 0 and 1.

Options: R&S ZNB-K20

Manual operation: See "Taps, Weight" on page 396

CALCulate<Chn>:EYE:EQUalization:DFE:MODE <Mode>

Defines if user-defined weights of the tap coefficients are used or if the weights are calculated automatically.

Suffix:
<Chn> The channel number used to identify the active trace.

Parameters:
<Mode> MANual | AUTomatic

MANual
The tap coefficient weights user-defined by `CALCulate<Chn>:EYE:EQUalization:DFE:LIST:TAPS` on page 940 are used.

AUTomatic
The tap coefficients are computed automatically.

Options: R&S ZNB-K20

Manual operation: See "Manual, Automatic" on page 396

CALCulate<Chn>:EYE:EQUalization:DFE:STATe <State>

Enables or disables DFE equalization.

Suffix:
<Chn> The channel number used to identify the active trace.

Parameters:
<State> ON | OFF - enables or disables DFE equalization.

Options: R&S ZNB-K20

Manual operation: See "DFE" on page 396

CALCulate<Chn>:EYE:EQUalization:DFE:TAP<Num> <Weight>

Defines the weight for the tap with the specified number.

Suffix:
<Chn> The channel number used to identify the active trace.

<Num> The number of the tap.

Parameters:
<Weight> Numeric value between 0 and 1

Options: R&S ZNB-K20

Manual operation: See "Taps, Weight" on page 396

CALCulate<Chn>:EYE:EQUalization:FFE:COUNt:POSTcursor <Count>

Defines the number of postcursors for FFE equalization.

Suffix:
<Chn> The channel number used to identify the active trace.

Parameters:
<Count> Numeric value

Options: R&S ZNB-K20

Manual operation: See "Postcurser Taps, Weight" on page 396

CALCulate<Chn>:EYE:EQUalization:FFE:COUNt:PRECursor <Count>

Defines the number of precursors for FFE equalization.

Suffix:
<Chn> The channel number used to identify the active trace.

Parameters:
<Count> Numeric value

Options: R&S ZNB-K20

Manual operation: See "Precurser Taps, Weight" on page 396

CALCulate<Chn>:EYE:EQUalization:FFE:LIST:POSTcursor <Weight>...

Lists the weights for the FFE postcursor taps.

Suffix:
<Chn> The channel number used to identify the active trace.

Parameters:
<Weight> List of numeric values between 0 and 1

Options: R&S ZNB-K20

Manual operation: See "Postcurser Taps, Weight" on page 396

CALCulate<Chn>:EYE:EQUalization:FFE:LIST:PRECursor <Weight>...

Lists the weights for the FFE precursor taps.

Suffix:
<Chn> The channel number used to identify the active trace.

Parameters:
<Weight> List of numeric values between 0 and 1

Options: R&S ZNB-K20

Manual operation: See "Precurser Taps, Weight" on page 396

CALCulate<Chn>:EYE:EQUalization:FFE:MAIN:TAP <Weight>...

Defines the weight for the postcursor tap with the specified number.

Suffix:
<Chn> The channel number used to identify the active trace.

Parameters:
<Weight> Numeric value between 0 and 1

Options: R&S ZNB-K20

Manual operation: See "Main Tap Weight" on page 396

CALCulate<Chn>:EYE:EQUalization:FFE:MODE <Mode>

Defines if user-defined weights of the tap coefficients are used or if the weights are calculated automatically.

Suffix:
<Chn> The channel number used to identify the active trace.

Parameters:
<Mode> MANual | AUTomatic

MANual
The user-defined tap coefficient weights are used.

AUTomatic
The tap coefficients are computed automatically.

Options: R&S ZNB-K20

Manual operation: See "Manual, Automatic" on page 395

CALCulate<Chn>:EYE:EQUalization:FFE:POSTcursor<Num> <Weight>

Defines the weight for the postcursor tap with the specified number.

Suffix:
<Chn> The channel number used to identify the active trace.

<Num> The number of the tap.

Parameters:
<Weight> Numeric value between 0 and 1

Options: R&S ZNB-K20

Manual operation: See "Postcurser Taps, Weight" on page 396

CALCulate<Chn>:EYE:EQUalization:FFE:PRECursor<Num> <Weight>

Defines the weight for the precursor tap with the specified number.

Suffix:
<Chn> The channel number used to identify the active trace.

<Num> The number of the tap.

Parameters:
<Weight> Numeric value between 0 and 1

Options: R&S ZNB-K20

Manual operation: See "Precurser Taps, Weight" on page 396

CALCulate<Chn>:EYE:EQUalization:FFE:STATe <State>

Enables or disables FFE equalization for the eye diagram.

Suffix:
<Chn> The channel number used to identify the active trace.

Parameters:
<State> ON | OFF - enables or disables equalization.

Options: R&S ZNB-K20

Manual operation: See "FFE" on page 395

CALCulate<Chn>:EYE:EQUalization:STATe <State>

Enables/disables the CTLE at the receiver simulation of the related eye diagram.

This command will raise an execution error if the active trace in the selected channel is not an eye diagram.

The parameters of the equalizer (a two-pole filter with single zero) can be specified using `CALCulate<Chn>:EYE:EQUalization:CTLE:DC`, `CALCulate<Chn>:EYE:EQUalization:CTLE:ZERO<Num>` and `CALCulate<Chn>:EYE:EQUalization:CTLE:POLE<Num>`.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<State> *RST: OFF

Options: R&S ZNB-K20

Manual operation: See "Equalization" on page 408

CALCulate<Chn>:EYE:INPut:BPATtern:TYPE <BitPattern>

Defines the type of bit stream to be simulated for the related eye diagram.

This command will raise an execution error if the active trace in the selected channel is not an eye diagram.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<BitPattern> PRBS | USER

PRBS
Pseudo-random binary sequence of selectable length (see `CALCulate<Chn>:EYE:INPut:LENGth:PRBS`)

USER
User-defined bit pattern, can be loaded from a 7-bit ASCII file (see `MMEMory:LOAD:EYE:BPATtern`) and is repeated until the configured bit length is reached (see `CALCulate<Chn>:EYE:INPut:LENGth:BITS`). If not loaded from file, the default pattern "10" is repeated.

Options: R&S ZNB-K20

Manual operation: See "Bit Stream" on page 384

CALCulate<Chn>:EYE:INPut:DRATe <Rate>

Defines the data rate of the bit stream generator for the related eye diagram.

This command will raise an execution error if the active trace in the selected channel is not an eye diagram.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<Rate> Data rate with units BPS, KBPS (=10^3 BPS), MBPS (=10^6 BPS), GBPS (=10^9 BPS)

Default unit: BPS

Options: R&S ZNB-K20

Manual operation: See "Symbol Rate/Transfer Rate" on page 385

CALCulate<Chn>:EYE:INPut:LENGth:BITS <BitLength>

Defines the length of a user-defined bit stream to be simulated for the related eye diagram. The user-defined bit pattern (the default "10" or loaded from file, see `MMEMory:LOAD:EYE:BPATtern`) is repeated until the specified length is reached.

This command will raise an execution error if the active trace in the selected channel is not an eye diagram.

Before executing this command, the pattern type must be set to USER (see CALCulate<Chn>:EYE:INPut:BPATtern:TYPE).

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<BitLength> Stream length, specified with units BITS, KIBITS (2^10 BITS), MIBITS (2^20 BITS), or GIBITS (2^30 BITS)
Default unit: BITS

Options: R&S ZNB-K20

Manual operation: See "Length" on page 384

CALCulate<Chn>:EYE:INPut:LENGth:PRBS <PrbsLength>

Defines the length of the pseudo-random binary sequence to be simulated for the related eye diagram.

This command will raise an execution error if the active trace in the selected channel is not an eye diagram.

Before executing this command, the pattern type must be set to PRBS (see CALCulate<Chn>:EYE:INPut:BPATtern:TYPE).

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<PrbsLength> L5 | L7 | L9 | L10 | L11 | L13 | L15 | L17 | L19 | L21 | L23 | L25 | L27 | L29 | L31
L*n* represents an actual sequence length of 2^n - 1.

Options: R&S ZNB-K20

Manual operation: See "Length" on page 384

CALCulate<Chn>:EYE:INPut:MODulation <PAMType>

Defines or queries the modulation type of the signal to be simulated for the related eye diagram.

This command will raise an execution error if the active trace in the selected channel is not an eye diagram.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<PAMType> NRZ | PM4 | PM8 | PM16
NRZ, PAM-4, PAM-8 or PAM-16
*RST: NRZ

Manual operation: See "Modulation" on page 385

CALCulate<Chn>:EYE:INPut:XRATe <Rate>

Defines the transfer rate of the bit stream generator for the related eye diagram.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<Rate> Transfer rate with units XPS (= transfers per second), KXPS (=10^3 XPS), MXPS (=10^6 XPS), GXPS (=10^9 XPS)
Default unit: XPS

Options: R&S ZNB-K20

Manual operation: See "Symbol Rate/Transfer Rate" on page 385

CALCulate<Chn>:EYE:INPut:OLEVel <OneLevel>
CALCulate<Chn>:EYE:INPut:ZLEVel <VoltageLevel>

Defines the highest/lowest (nominal) voltage level of the simulated digital signal generating the related eye diagram.

This command will raise an execution error if the active trace in the selected channel is not an eye diagram.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<VoltageLevel> Default unit: V

Options: R&S ZNB-K20

Manual operation: See "High Level / Low Level" on page 386

CALCulate<Chn>:EYE:INPut:RTIMe:DATA <RiseTime>

Sets/gets the rise time of the low pass in the binary signal generator simulation of the related eye diagram.

This command will raise an execution error if the active trace in the selected channel is not an eye diagram.

The low pass is enabled using `CALCulate<Chn>:EYE:STIMulus:LOWPass`. The rise time definition can be modified using `CALCulate<Chn>:EYE:INPut:RTIMe:THReshold`.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<RiseTime> Default unit: s

Options: R&S ZNB-K20

Manual operation: See "Rise Time / Rise Time Definition" on page 386

CALCulate<Chn>:EYE:INPut:RTIMe:THReshold <RiseThreshold>

Selects the appropriate rise time definition for the low pass in the binary signal generator simulation of the related eye diagram.

The rise time itself can be configured using `CALCulate<Chn>:EYE:INPut:RTIMe:DATA`.

This command will raise an execution error if the active trace in the selected channel is not an eye diagram.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<RiseThreshold> T1_9 | T2_8

T1_9: 10% to 90% (of the voltage amplitude between 1-level and 0-level)
Default threshold for eye diagrams.
T2_8: 20% to 80%

Options: R&S ZNB-K20

Manual operation: See "Rise Time / Rise Time Definition" on page 386

CALCulate<Chn>:EYE:JITTer:DIRac:DELTa <DiracDelta>

Defines the magnitude of the Dirac jitter in the generator simulation of the related eye diagram.

This command will raise an execution error if the active trace in the selected channel is not an eye diagram.

Dirac jitter insertion can be enabled using `CALCulate<Chn>:EYE:JITTer:TYPE:PERiodic`.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<DiracDelta> *RST: 1 ns
Default unit: s

Options: R&S ZNB-K20

Manual operation: See "Dirac" on page 391

CALCulate<Chn>:EYE:JITTer:DIRac:PROBability <DiracProbability>

Defines the probability of the Dirac jitter in the generator simulation of the related eye diagram.

This command will raise an execution error if the active trace in the selected channel is not an eye diagram.

Dirac jitter insertion can be enabled using `CALCulate<Chn>:EYE:JITTer:TYPE:PERiodic`.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<DiracProbability> Jitter probability at each symbol period
*RST: 0.5

Options: R&S ZNB-K20

Manual operation: See "Dirac" on page 391

CALCulate<Chn>:EYE:JITTer:PERiodic:FREQuency <PeriodicFrequency>

Defines the frequency of the periodic jitter in the generator simulation of the related eye diagram.

This command will raise an execution error if the active trace in the selected channel is not an eye diagram.

Periodic jitter insertion can be enabled using `CALCulate<Chn>:EYE:JITTer:TYPE:PERiodic`.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<PeriodicFrequency> *RST: 10 MHz
Default unit: Hz

Options: R&S ZNB-K20

Manual operation: See "Periodic" on page 391

CALCulate<Chn>:EYE:JITTer:PERiodic:MAGNitude <PeriodicMagnitude>

Defines the magnitude of the periodic jitter in the generator simulation of the related eye diagram.

This command will raise an execution error if the active trace in the selected channel is not an eye diagram.

Periodic jitter insertion can be enabled using `CALCulate<Chn>:EYE:JITTer:TYPE:PERiodic`.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<PeriodicMagnitude> *RST: 1 ns
Default unit: s

Options: R&S ZNB-K20

Manual operation: See "Periodic" on page 391

CALCulate<Chn>:EYE:JITTer:PERiodic:PHASe <PeriodicPhase>

Defines the phase of the periodic jitter in the generator simulation of the related eye diagram.

This command will raise an execution error if the active trace in the selected channel is not an eye diagram.

Periodic jitter insertion can be enabled using `CALCulate<Chn>:EYE:JITTer:TYPE:PERiodic`.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<PeriodicPhase> *RST: 0 deg
Default unit: deg

Options: R&S ZNB-K20

Manual operation: See "Periodic" on page 391

CALCulate<Chn>:EYE:JITTer:RANDom:STDDeviation <StdDeviation>

Defines the standard deviation of the random jitter in the generator simulation of the related eye diagram.

This command will raise an execution error if the active trace in the selected channel is not an eye diagram.

Random jitter insertion can be enabled using `CALCulate<Chn>:EYE:JITTer:TYPE:RANDom`.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<StdDeviation> Default unit: s

Options: R&S ZNB-K20

Manual operation: See "Random" on page 391

CALCulate<Chn>:EYE:JITTer:STATe <Boolean>

Activates the jitter functionality in the generator simulation of the related eye diagram.

This command will raise an execution error if the active trace in the selected channel is not an eye diagram.

The individual jitter sources can be selectively enabled using

- `CALCulate<Chn>:EYE:JITTer:TYPE:DIRac`
- `CALCulate<Chn>:EYE:JITTer:TYPE:PERiodic`

- CALCulate<Chn>:EYE:JITTer:TYPE:RANDom
- CALCulate<Chn>:EYE:JITTer:TYPE:USER

This command will raise an execution error if the active trace in the selected channel is not an eye diagram.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<Boolean> *RST: OFF

Options: R&S ZNB-K20

Manual operation: See "Active" on page 391

CALCulate<Chn>:EYE:JITTer:TYPE:DIRac <Boolean>

Enables/disables Dirac jitter insertion in the generator simulation of the related eye diagram.

This command will raise an execution error if the active trace in the selected channel is not an eye diagram.

Amplitude and probability of the jitter can be specified using CALCulate<Chn>:EYE:JITTer:DIRac:DELTa and CALCulate<Chn>:EYE:JITTer:DIRac:PROBability, respectively.

Note that jitter is not applied unless the jitter functionality is activated using CALCulate<Chn>:EYE:JITTer:STATe.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<Boolean> *RST: OFF

Options: R&S ZNB-K20

Manual operation: See "Dirac" on page 391

CALCulate<Chn>:EYE:JITTer:TYPE:PERiodic <Boolean>

Enables/disables periodic jitter insertion in the generator simulation of the related eye diagram.

This command will raise an execution error if the active trace in the selected channel is not an eye diagram.

Amplitude, frequency and phase of the jitter can be specified using CALCulate<Chn>:EYE:JITTer:PERiodic:MAGNitude, CALCulate<Chn>:EYE:JITTer:PERiodic:FREQuency and CALCulate<Chn>:EYE:JITTer:PERiodic:PHASe, respectively.

Note that jitter is not applied unless the jitter functionality is activated using CALCulate<Chn>:EYE:JITTer:STATe.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<Boolean> *RST: OFF

Options: R&S ZNB-K20

Manual operation: See "Periodic" on page 391

CALCulate<Chn>:EYE:JITTer:TYPE:RANDom <Boolean>

Enables/disables random jitter insertion in the generator simulation of the related eye diagram.

This command will raise an execution error if the active trace in the selected channel is not an eye diagram.

The standard deviation can be set using `CALCulate<Chn>:EYE:JITTer:RANDom:STDDeviation`.

Note that jitter is not applied unless the jitter functionality is activated using `CALCulate<Chn>:EYE:JITTer:STATe`.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<Boolean> *RST: OFF

Options: R&S ZNB-K20

Manual operation: See "Random" on page 391

CALCulate<Chn>:EYE:JITTer:TYPE:USER <Boolean>

Enables/disables user-defined jitter insertion in the generator simulation of the related eye diagram.

This command will raise an execution error if the active trace in the selected channel is not an eye diagram.

Before enabling user-defined jitter, the jitter values must have been loaded from file using `MMEMory:LOAD:EYE:JITTer`.

Note that jitter is not applied unless the jitter functionality is activated using `CALCulate<Chn>:EYE:JITTer:STATe`.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<Boolean> *RST: OFF

Options: R&S ZNB-K20

Manual operation: See "User Specific" on page 392

CALCulate<Chn>:EYE:MASK AUTO

Automatically creates an eye mask, based on the current eye measurement settings.

Suffix:
<Chn> Channel number used to identify the active trace

Usage: Setting only

Manual operation: See "Automatic Mask Generation" on page 402

CALCulate<Chn>:EYE:MASK:CENTer:HORizontal <HorizontalOffset>

Defines the horizontal center of the eye mask in the related eye diagram.

This command will raise an execution error if the active trace in the selected channel is not an eye diagram.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<HorizontalOffset> Horizontal offset relative to the middle of the eye diagram
Default unit: s

Options: R&S ZNB-K20

Manual operation: See "Mask Center" on page 401

CALCulate<Chn>:EYE:MASK:CENTer:VERTical <VerticalOffset>

Defines the vertical center of the eye mask in the related eye diagram.

This command will raise an execution error if the active trace in the selected channel is not an eye diagram.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<VerticalOffset> Vertical offset relative to the 0 V level, i.e. the Voltage level of the center.
Default unit: V

Options: R&S ZNB-K20

Manual operation: See "Mask Center" on page 401

CALCulate<Chn>:EYE:MASK:DATA?

Returns the detailed results of the mask test in the related eye diagram (i.e. the contents of the corresponding result info field).

This command raises an execution error if the active trace in the selected channel is not an eye diagram, or if `CALCulate<Chn>:EYE:MASK:STATe` is `OFF`.

Use MMEMory:STORe:EYE:MASK:RESults to save these results to an ASCII file.

Suffix:
<Chn> Channel number used to identify the active trace

Example: CALCulate1:EYE:MASK:DATA?
Returns something like

```
'Eye Mask
-----------------------------
Fail Condition Type Samples
Violation Tolerance 1
Total Number of Samples 10342

Mask 1 (Top) Active

Samples Hits 366
Fail Rate 3.539 %
Test Result Fail

Mask 2 (Bottom) Not Active

Samples Hits ------
Fail Rate ------
Test Result ------

Mask 3 (Octagon) Not Active

Samples Hits ------
Fail Rate ------
Test Result ------

'
```

Usage: Query only

Options: R&S ZNB-K20

Manual operation: See "Mask Test On" on page 397

CALCulate<Chn>:EYE:MASK:FAIL?

Returns 'Pass' or 'Fail' to indicate the result of the limit check in the related eye diagram.

This command raises e an execution error if the active trace in the selected channel is not an eye diagram, if the eye test is not enabled (CALCulate<Chn>:EYE:MASK:STATe ON), or if the mask is empty (see CALCulate<Chn>:EYE:MASK:SHAPe:BOTTom|TOP|POLYgon:STATe.

Tip

- Use CALCulate:CLIMits:FAIL? to perform a composite (global) limit check

- The result is automatically recalculated whenever a relevant setting is changed, i.e. a subsequent query returns the updated mask test result

Suffix:
<Chn> Channel number used to identify the active trace

Return values:
<TestResults>

Usage: Query only

Options: R&S ZNB-K20

Manual operation: See "Mask Test On" on page 397

CALCulate<Chn>:EYE:MASK:FAIL:BEEP <Boolean>

Deprecated command: Since version 3.45, generating sounds is no longer supported by the analyzer firmware.

Defines whether the R&S ZNB makes an audible beep on mask failures in the related eye diagram.

This command will raise an execution error if the active trace in the selected channel is not an eye diagram.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<Boolean> *RST: OFF

Options: R&S ZNB-K20

CALCulate<Chn>:EYE:MASK:FAIL:CONDition <FailCondition>

Defines whether the fail condition for the eye mask test of the related eye diagram is specified in absolute or relative terms.

This command will raise an execution error if the active trace in the selected channel is not an eye diagram.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<FailCondition> SAMPles | RATE

SAMPles: The eye mask test fails if a configurable number of samples violate the mask (see `CALCulate<Chn>:EYE:MASK:VIOLation:TOLerance`).
RATE: The eye mask test fails if the share of the samples violating the mask is higher than a configurable percentage (see `CALCulate<Chn>:EYE:MASK:VIOLation:RATE`).

*RST: SAMPles

Options: R&S ZNB-K20

Manual operation: See "Test Settings" on page 401

CALCulate<Chn>:EYE:MASK:SHAPe:BOTTom:HORizontal <Width>
CALCulate<Chn>:EYE:MASK:SHAPe:TOP:HORizontal <Width>

Defines the width of the bottom/top rectangle in the eye mask of the related eye diagram.

This command will raise an execution error if the active trace in the selected channel is not an eye diagram.

Use `CALCulate<Chn>:EYE:MASK:SHAPe:BOTTom:STATe` or `CALCulate<Chn>:EYE:MASK:SHAPe:TOP:STATe` to activate or deactivate the area in the eye mask. The eye mask test is enabled/disabled using `CALCulate<Chn>:EYE:MASK:STATe`.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<Width> Default unit: s

Options: R&S ZNB-K20

Manual operation: See "Top/Bottom Setup" on page 400

CALCulate<Chn>:EYE:MASK:SHAPe:BOTTom:STATe <Boolean>
CALCulate<Chn>:EYE:MASK:SHAPe:POLYgon:STATe <Boolean>
CALCulate<Chn>:EYE:MASK:SHAPe:TOP:STATe <Boolean>

Activates/deactivates the respective area in the eye mask of the related eye diagram.

This command will raise an execution error if the active trace in the selected channel is not an eye diagram.

The eye mask test is enabled/disabled using `CALCulate<Chn>:EYE:MASK:STATe`.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<Boolean> *RST: OFF

Options: R&S ZNB-K20

Manual operation: See "Top/Bottom Setup" on page 400

CALCulate<Chn>:EYE:MASK:SHAPe:BOTTom:VERTical <Offset>
CALCulate<Chn>:EYE:MASK:SHAPe:TOP:VERTical <Offset>

Defines the offset of the bottom/top rectangle in the mask of the related eye diagram. The offset is specified relative to the vertical center of the eye mask (see `CALCulate<Chn>:EYE:MASK:CENTer:VERTical` on page 952).

This command will raise an execution error if the active trace in the selected channel is not an eye diagram.

Use `CALCulate<Chn>:EYE:MASK:SHAPe:BOTTom:STATe` or `CALCulate<Chn>:EYE:MASK:SHAPe:TOP:STATe` to activate or deactivate the area in the eye mask. The eye mask test is enabled/disabled using `CALCulate<Chn>:EYE:MASK:STATe`.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<Offset> Default unit: V

Options: R&S ZNB-K20

Manual operation: See "Top/Bottom Setup" on page 400

CALCulate<Chn>:EYE:MEASurement:BASic <Boolean>
CALCulate<Chn>:EYE:MEASurement:TIME <Boolean>

Selectively show or hide "Basic" and "Time" measurement results in the eye measurements info field, if display of measurement results is enabled using `CALCulate<Chn>:EYE:MEASurement:STATe ON`).

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<Boolean>
ON (1)
Show basic/time results
OFF (0)
Hide basic/time results
*RST: ON (1)

Options: R&S ZNB-K20

Manual operation: See "Measurements..." on page 382

CALCulate<Chn>:EYE:MASK:SHAPe:POLYgon:HORizontal <Main>[, <Minor>]

Defines the main [and minor] width of the center polygon in the mask of the related eye diagram. The geometric interpretation depends on the selected polygon type (see `CALCulate<Chn>:EYE:MASK:SHAPe:POLYgon:TYPE` on page 957):

This command will raise an execution error if the active trace in the selected channel is not an eye diagram.

Use `CALCulate<Chn>:EYE:MASK:SHAPe:POLYgon:STATe` to activate or deactivate the center polygon in the eye mask. The eye mask test is enabled/disabled using `CALCulate<Chn>:EYE:MASK:STATe`.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:

<Main> Default unit: s

<Minor> Default unit: s

Options: R&S ZNB-K20

Manual operation: See "Polygon Setup" on page 399

CALCulate<Chn>:EYE:MASK:SHAPe:POLYgon:TYPE <PolygonType>

Defines the shape of the center polygon (octagon, hexagon or rectangle) in the eye mask of the related eye diagram.

This command will raise an execution error if the active trace in the selected channel is not an eye diagram.

Use `CALCulate<Chn>:EYE:MASK:SHAPe:POLYgon:STATe` to activate or deactivate the area in the eye mask. The eye mask test is enabled/disabled using `CALCulate<Chn>:EYE:MASK:STATe`.

Suffix:

<Chn> Channel number used to identify the active trace

Parameters:

<PolygonType> OCTogon | HEXagon | RECTangle

Options: R&S ZNB-K20

Manual operation: See "Polygon Setup" on page 399

CALCulate<Chn>:EYE:MASK:SHAPe:POLYgon:VERTical <Main>[, <Minor>]

Defines the main [and minor] height of the center polygon in the mask of the related eye diagram. The geometric interpretation depends on the selected polygon type (see `CALCulate<Chn>:EYE:MASK:SHAPe:POLYgon:TYPE` on page 957):

This command will raise an execution error if the active trace in the selected channel is not an eye diagram.

Use `CALCulate<Chn>:EYE:MASK:SHAPe:POLYgon:STATe` to activate or deactivate the center polygon in the eye mask. The eye mask test is enabled/disabled using `CALCulate<Chn>:EYE:MASK:STATe`.

Suffix:

<Chn> Channel number used to identify the active trace

Parameters:

<Main> Default unit: V

<Minor> Default unit: V

Options: R&S ZNB-K20

Manual operation: See "Polygon Setup" on page 399

CALCulate<Chn>:EYE:MASK:SHOW <Boolean>

Defines the visibility of the configured eye mask in the related eye diagram.

This command will raise an execution error if the active trace in the selected channel is not an eye diagram.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<Boolean> *RST: OFF

Options: R&S ZNB-K20

Manual operation: See "Show Mask" on page 397

CALCulate<Chn>:EYE:MASK:STATe <Boolean>

Defines whether the eye mask test should be run after every recalculation of the related eye diagram.

This command will raise an execution error if the active trace in the selected channel is not an eye diagram.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<Boolean> *RST: OFF

Options: R&S ZNB-K20

Manual operation: See "Mask Test On" on page 397

CALCulate<Chn>:EYE:MASK:VIOLation:RATE <ViolationRate>

Defines the violation rate (i.e. the share of bad samples) for the mask test in the related eye diagram.

This command will raise an execution error if the active trace in the selected channel is not an eye diagram.

This rate is only used if `CALCulate<Chn>:EYE:MASK:FAIL:CONDition` is set to RATE.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<ViolationRate> Default unit: percent

Options: R&S ZNB-K20

Manual operation: See "Test Settings" on page 401

CALCulate<Chn>:EYE:MASK:VIOLation:TOLerance <ViolationTolerance>

Defines the violation tolerance (i.e. the number of bad samples) for the mask test in the related eye diagram.

This command will raise an execution error if the active trace in the selected channel is not an eye diagram.

This tolerance is only used if `CALCulate<Chn>:EYE:MASK:FAIL:CONDition` is set to SAMPles.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<ViolationTolerance>

Options: R&S ZNB-K20

Manual operation: See "Test Settings" on page 401

CALCulate<Chn>:EYE:MEASurement:DATA?

Returns the measurement results of the related eye diagram (see "Eye diagram results" on page 227).

The return value is of type string and returns the eye measurement results in csv format with decimal separator "." and field separator "," (see also `MMEMory:STORe:EYE:MEASurements`).

Note that the full set of measurement results is only available for NRZ modulated generator signals (see `CALCulate<Chn>:EYE:INPut:MODulation` on page 945).

This command will raise an execution error if the active trace in the selected channel is not an eye diagram.

Suffix:
<Chn> Channel number used to identify the active trace

Example: For NRZ modulated generator signals, `CALCulate:EYE:MEASurement:DATA?` returns something like

```
'Eye Measurements
Eye Minimum,-102.344 mV,
Eye Maximum,2.699 V,
Eye Base,277.189 µV,
Eye Top,2.597 V,
Eye Mean,1.298 V,
Eye Amplitude,2.596 V,
Eye Height,2.596 V,
Eye Width,10.000 ns,
Bit Period,10.000 ns,
Rise Time,115.000 ps,
Fall Time,115.000 ps,
Jitter Pk-Pk,50.125 ps,
Jitter RMS,0.000 s,
Duty Cycle Dist,0.000 s,
Duty Cycle Pct,0.000 %,
Crossing Percent,50.000 %,
Opening Factor,1.000 ,
SNR,0.000 '
```

For PAM modulated generator signals a reduced set of measurement results is returned:

```
'Eye Measurements
Eye Minimum,-2.909 V,
Eye Maximum,2.997 V,
Eye Base,-2.734 V,
Eye Top,2.734 V,
Eye Mean,0.000 V,
Eye Amplitude,5.468 V'
```

Usage: Query only

Options: R&S ZNB-K20

Manual operation: See "Display Measurements" on page 381

CALCulate<Chn>:EYE:MEASurement:STATe <Boolean>

Defines the visibility of the result info field in the related eye diagram.

This command will raise an execution error if the active trace in the selected channel is not an eye diagram.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<Boolean>

Options: R&S ZNB-K20

Manual operation: See "Display Measurements" on page 381

CALCulate<Chn>:EYE:MEASurement:TTIMe:THReshold <ThresholdEnum|LowerThreshold>[, <UpperThreshold>]

Defines the lower and upper thresholds that are used to calculate the transition times (rise/fall time) in an eye measurement.

The thresholds can either be specified by enum constants for the standard 10–90% or 20–80% rise times, or numerically.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<ThresholdEnum|LowerThreshold> T1_9: 10–90% rise time
T2_8: 20–80% rise time
<integer>: Lower rise time threshold as integer percentage
*RST: T1_9:

<UpperThreshold> Upper rise time threshold as integer percentage

Options: R&S ZNB-K20

Manual operation: See "Measurements..." on page 382

CALCulate<Chn>:EYE:NOISe:RMS <NoiseRMS>

Defines the root mean square (RMS) noise level in the generator simulation of the related eye diagram.

This command will raise an execution error if the active trace in the selected channel is not an eye diagram.

Gaussian noise insertion can be enabled using `CALCulate<Chn>:EYE:NOISe:STATe`.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<NoiseRMS> Default unit: V

Manual operation: See "RMS" on page 393

CALCulate<Chn>:EYE:NOISe:STATe <Boolean>

Enables/disables Gaussian noise in the generator simulation of the related eye diagram.

This command will raise an execution error if the active trace in the selected channel is not an eye diagram.

The noise RMS level can be set using `CALCulate<Chn>:EYE:NOISe:RMS`.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<Boolean> *RST: OFF

Options: R&S ZNB-K20

Manual operation: See "Active" on page 393

CALCulate<Chn>:EYE:STATe <Boolean>

Defines whether the active trace in the selected channel shall be represented as an eye diagram.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<Boolean>

Example:
```
*RST
// switch trace to to S31 and switch on the eye diagram
:CALCulate1:PARameter:MEASure 'Trc1', 'S31'
CALCulate1:EYE:STATe ON
```

Options: R&S ZNB-K20

Manual operation: See "Eye Diagram" on page 381

CALCulate<Chn>:EYE:STIMulus:ENCoder <Boolean>

Enables or disables 8b/10b encoding in the bit stream simulation of the related eye diagram.

This command will raise an execution error if the active trace in the selected channel is not an eye diagram.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<Boolean> *RST: OFF

Options: R&S ZNB-K20

Manual operation: See "Encoder" on page 388

CALCulate<Chn>:EYE:STIMulus:LOWPass <Boolean>

Enables/disables a single pole low pass filter in the binary signal generator simulation of the related eye diagram measurement.

The low-pass is defined using its rise time (see `CALCulate<Chn>:EYE:INPut:RTIMe:DATA` on page 946) and rise time definition (see `CALCulate<Chn>:EYE:INPut:RTIMe:THReshold` on page 947).

This command will raise an execution error if the active trace in the selected channel is not an eye diagram.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<Boolean> *RST: OFF

Options: R&S ZNB-K20

Manual operation: See "Low Pass" on page 386

CALCulate<Chn>:EYE:STIMulus:SCRambler <Boolean>

Enables/disables the scrambler in the bit stream simulation of the related eye diagram.

This command will raise an execution error if the active trace in the selected channel is not an eye diagram.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<Boolean> *RST: OFF

Options: R&S ZNB-K20

Manual operation: See "Scrambler" on page 388

CALCulate<Chn>:EYE:VIEW <EyeView>

Allows you to shorten the calculation chain of the related eye diagram without deactivating the building blocks at the tail end.

This command will raise an execution error if the active trace in the selected channel is not an eye diagram.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:

<EyeView> STIMulus | EMPHasis | JITTer | NOISe | DUT | EQUalization

The rightmost building block to be included the calculation chain STIMulus > EMPHasis > JITTer > NOISe > DUT > EQUalization. Note that the selected block must be active, for example `CALCulate1:EYE:VIEW JITTER` requires `CALCulate<Chn>:EYE:JITTer:STATe` to be `ON`. Conversely, if `CALCulate1:EYE:VIEW` is set to `JITTER` and then `CALCulate1:EYE:JITTer:STATe` is turned `OFF`, the calculation chain is further shortened to the next active building block.

Options: R&S ZNB-K20

Manual operation: See "[Slider]" on page 387

7.3.1.5 CALCulate:FILTer[:GATE]...

The `CALCulate:FILTer[:GATE]...` commands define the properties of the time gate which is used to optimize the time domain response.

CALCulate<Chn>:FILTer[:GATE]:TIME:ADD 964
CALCulate:FILTer[:GATE]:TIME:AOFFset 965
CALCulate<Chn>:FILTer[:GATE]:TIME:CATalog? 965
CALCulate<Chn>:FILTer[:GATE]:TIME:CENTer 965
CALCulate<Chn>:FILTer[:GATE]:TIME:DCHebyshev 966
CALCulate<Chn>:FILTer[:GATE]:TIME:DELete 966
CALCulate<Chn>:FILTer[:GATE]:TIME:LINK 967
CALCulate<Chn>:FILTer[:GATE]:TIME:SELect 967
CALCulate<Chn>:FILTer[:GATE]:TIME:SHAPe 967
CALCulate<Chn>:FILTer[:GATE]:TIME:SHOW 968
CALCulate<Chn>:FILTer[:GATE]:TIME:SPAN 968
CALCulate<Chn>:FILTer[:GATE]:TIME:STARt 969
CALCulate<Chn>:FILTer[:GATE]:TIME:STOP 969
CALCulate<Chn>:FILTer[:GATE]:TIME:STATe 969
CALCulate<Chn>:FILTer[:GATE]:TIME[:TYPE] 970
CALCulate<Chn>:FILTer[:GATE]:TIME:WINDow 970

CALCulate<Chn>:FILTer[:GATE]:TIME:ADD

Adds a time gate to channel <Chn>'s active trace and selects it for configuration purposes (see `CALCulate<Chn>:FILTer[:GATE]:TIME:SELect`.

For each trace, time gates are named 'TG<n>' with positive integers <n>. Whenever you `ADD` a time gate, the highest available <n> is incremented by 1.

Use `CALCulate<Chn>:FILTer[:GATE]:TIME:DELete` to delete the selected time gate.

Suffix:

<Chn> Channel number used to identify the active trace

Usage: Event

Manual operation: See "Add Time Gate" on page 364

CALCulate:FILTer[:GATE]:TIME:AOFFset <Boolean>

Activates the operating mode where the time gate is moved in the opposite direction when the "Delay" setting is changed.

Parameters:

<Boolean> ON | OFF - enable or disable "Adjust Time Gate".

*RST: OFF

Example:
```
*RST; :CALCulate1:TRANsform:TIME:STATe ON
CALCulate1:FILTer:GATE:TIME:STATe ON; SHOW ON
```
Activate time domain representation and a time gate in channel no. 1. Display the time gate
```
CALCulate1:FILTer:GATE:TIME:STARt 2ns; STOP 3 ns
```
Restrict the time gate to the time interval between 2 ns and 3 ns.
```
CALCulate:FILTer:GATE:TIME:AOFFset ON
```
Activate an offset of the time gate according to a new delay setting.
```
SENSe1:CORRection:EDELay1:TIME 1ns
```
Specify a 1 ns delay at port 1.
```
CALCulate1:FILTer:GATE:TIME:STARt?; STOP?
```
Query the time gate position. The response is `1E-009;2E-009`.

Manual operation: See "Adjust Time Gate" on page 739

CALCulate<Chn>:FILTer[:GATE]:TIME:CATalog?

Queries for existing time gate names of channel <Chn>'s active trace.

For each trace, time gates are named 'TG<n>' with positive integers <n>, not necessarily consecutive. Whenever you add a time gate using `CALCulate<Chn>:FILTer[:GATE]:TIME:ADD`, the highest available <n> is incremented by 1.

Suffix:

<Chn> Channel number used to identify the active trace

Usage: Query only

Manual operation: See "Selected Time Gate" on page 364

CALCulate<Chn>:FILTer[:GATE]:TIME:CENTer <CenterTime>

This command applies to the selected time gate of channel <Chn>'s active trace (see `CALCulate<Chn>:FILTer[:GATE]:TIME:SELect`).

It defines the center time of this time gate.

Use `CALCulate<Chn>:FILTer[:GATE]:TIME:SPAN` to set its span.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<CenterTime> Center time of the time gate

Range: -99.8999999 s to +99.8999999 s
Increment: 0.1 ns
*RST: 1.5E-009 s
Default unit: s

Example: `*RST; :CALC:TRAN:TIME:STAT ON; :CALC:FILT:TIME:STAT ON`
Reset the instrument and enable the time domain representation and the time gate.
`CALC:FILT:TIME:CENT 0; SPAN 5ns`
Set the center time to 0 ns and the time span to 5 ns.

Manual operation: See "Axis Pair" on page 364

CALCulate<Chn>:FILTer[:GATE]:TIME:DCHebyshev <SidebandSupp>

Sets the sideband suppression of a Dolph-Chebyshev time gate.

This command applies to the selected time gate of channel <Chn>'s active trace (see `CALCulate<Chn>:FILTer[:GATE]:TIME:SELect`).

It is only available if the window shape of this time gate is Dolph-Chebyshev (`CALCulate<Chn>:FILTer[:GATE]:TIME:WINDow DCHebyshev`).

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<SidebandSupp> Sideband suppression

Range: 10 dB to 120 dB
Increment: 10 dB
*RST: 32 dB
Default unit: dB

Example: `*RST; :CALC:FILT:TIME:WIND DCH`
Reset the instrument and select a Dolph-Chebyshev time gate for filtering the data in the frequency domain.
`CALC:FILT:TIME:DCH 25`
Set the sideband suppression to 25 dB.

Manual operation: See "Side Lobe Level" on page 365

CALCulate<Chn>:FILTer[:GATE]:TIME:DELete <TGName>

Deletes the selected time gate of channel <Chn>'s active trace.

Use `CALCulate<Chn>:FILTer[:GATE]:TIME:ADD` to add a time gate to a trace and select it.

Suffix:
<Chn> Channel number used to identify the active trace

Setting parameters:
<TGName> Time gate name
Use `CALCulate<Chn>:FILTer[:GATE]:TIME:CATalog?` to query for existing time gate names.

Usage: Setting only

Manual operation: See "Delete Time Gate" on page 364

CALCulate<Chn>:FILTer[:GATE]:TIME:LINK <TraceName>

Allows you to link the time gates of different (time domain) traces, i.e. to align the time gate settings across linked (~coupled) traces.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<TraceName> Name of the trace to be linked to the active trace of channel <Chn>.
Use the empty string to unlink the active trace of channel <Chn>.

Manual operation: See "Time Gate Linking" on page 365

CALCulate<Chn>:FILTer[:GATE]:TIME:SELect <TGName>

Selects a time gate of channel <Chn>'s active trace for configuration purposes.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<TGName> Time gate name
Use `CALCulate<Chn>:FILTer[:GATE]:TIME:CATalog?` to query the existing time gate names.

Manual operation: See "Selected Time Gate" on page 364

CALCulate<Chn>:FILTer[:GATE]:TIME:SHAPe <TimeGate>

This command applies to the selected time gate of channel <Chn>'s active trace (see `CALCulate<Chn>:FILTer[:GATE]:TIME:SELect`).

It specifies the window shape of this time gate.

Tip:

Use the more general command `CALCulate<Chn>:FILTer[:GATE]:TIME:WINDow` if you wish to select a Dolph-Chebyshev time gate.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<TimeGate> MAXimum | WIDE | NORMal | MINimum

MINimum - Steepest edges (rectangle)
WIDE - Normal gate (Hann)
NORM - Steep edges (Hamming)
Maximum - Maximum flatness (Bohman)

*RST: WIDE

Example: `*RST; :CALC:FILT:TIME:SHAP?`
Reset the instrument and query the type of time gate used. The response is WIDE.

Manual operation: See "Shape" on page 365

CALCulate<Chn>:FILTer[:GATE]:TIME:SHOW <Boolean>

This command applies to all time gates of channel <Chn>'s active trace.

It enables or disables permanent display of its gate limits in a time domain diagram.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<Boolean> **OFF (0)**
Gate limits hidden

ON (1)
Gate limits permanently displayed

*RST: OFF

Example: See `CALCulate<Chn>:FILTer[:GATE]:TIME:CENTer`

Manual operation: See "Show Range Lines" on page 364

CALCulate<Chn>:FILTer[:GATE]:TIME:SPAN <Span>

This command applies to the selected time gate of channel <Chn>'s active trace (see `CALCulate<Chn>:FILTer[:GATE]:TIME:SELect`).

It defines the span of this time gate.

Use `CALCulate<Chn>:FILTer[:GATE]:TIME:CENTer` to set its center.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:

<Span> Span of the time gate.

Range: 2E-012 s to 200 s
Increment: 0.1 ns
*RST: 5E-009 s
Default unit: s

Example: See `CALCulate<Chn>:FILTer[:GATE]:TIME:CENTer`

Manual operation: See "Gate Span" on page 360

CALCulate<Chn>:FILTer[:GATE]:TIME:STARt <StartTime>
CALCulate<Chn>:FILTer[:GATE]:TIME:STOP <StopTime>

These commands apply to the selected time gate of channel <Chn>'s active trace (see `CALCulate<Chn>:FILTer[:GATE]:TIME:SELect`).

They define the start and stop time of the time gate, respectively.

Suffix:

<Chn> Channel number used to identify the active trace

Parameters:

<StopTime> Start or stop time of the time gate.

Range: -100 s to +99.999999999998 s (start time) and -99.999999999998 s to +100 s (stop time)
Increment: 0.1 ns
*RST: -1E-009 s (start time) to +4E-009 s (stop time)
Default unit: s

Example: `*RST; :CALC:TRAN:TIME:STAT ON; :CALC:FILT:TIME:STAT ON`
Reset the instrument and enable the time domain representation and the time gate.
`CALC:FILT:TIME:STAR 0; STOP 10ns; SHOW ON`
Set the start time to 0 ns and the stop time to 10 ns and display the time gate permanently.

Manual operation: See "Axis Pair" on page 364

Note: If the start frequency entered is greater than the current stop frequency, the stop frequency is set to the start frequency plus the minimum frequency span (`CALCulate<Chn>:FILTer[:GATE]:TIME:SPAN`).

If the stop frequency entered is smaller than the current start frequency, the start frequency is set to the stop frequency minus the minimum frequency span.

CALCulate<Chn>:FILTer[:GATE]:TIME:STATe <Boolean>

Enables the time gates of channel <Chn>'s active trace or disables them.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<Boolean> **OFF (0)**
Time gating disabled

ON (1)
Time gating enabled

*RST: OFF

Example: `*RST; :CALC:TRAN:TIME:STAT?`
`CALC:FILT:TIME:STAT?`
Reset the instrument, activating a frequency sweep, and query whether the default trace is displayed in the time domain and whether the time gate is enabled. The response to both queries is `0`.

Manual operation: See "Time Gate" on page 364

CALCulate<Chn>:FILTer[:GATE]:TIME[:TYPE] <TimeGateFilter>

This command applies to the selected time gate of channel <Chn>'s active trace (see `CALCulate<Chn>:FILTer[:GATE]:TIME:SELect`).

It selects this time gate's filter type, defining what occurs to the data in the specific time region.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<TimeGateFilter> BPASs | NOTCh

BPASs
Bandpass filter: Pass all information in the specified time region and reject everything else.

NOTCh
Notch filter: Reject all information in the specified time region and pass everything else.

*RST: BPASs

Example: `*RST; :CALC:FILT:TIME:STAT ON`
Reset the instrument and enable the time gate.
`CALC:FILT:TIME NOTCh`
Select a notch filter to reject unwanted pulses.

Manual operation: See "Type" on page 365

CALCulate<Chn>:FILTer[:GATE]:TIME:WINDow <TimeGate>

This command applies to the selected time gate of channel <Chn>'s active trace (see `CALCulate<Chn>:FILTer[:GATE]:TIME:SELect`).

It specifies the window shape of this time gate.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<TimeGate> RECT | HAMMing | HANNing | BOHMan | DCHebyshev

RECT
Steepest edges (rectangle)

HANN
Normal gate (Hann)

HAMMing
Steep edges (Hamming)

BOHMan
Minimum flatness (Bohman)

DCHebyshev
Arbitrary gate shape (Dolph-Chebyshev)

*RST: HANN
See Section 4.7.1.5, "Time gates", on page 223.

Example: See `CALCulate<Chn>:FILTer[:GATE]:TIME:DCHebyshev`

Manual operation: See "Shape" on page 365

7.3.1.6 CALCulate:FMODel...

The `CALCulate:FMODel...` commands are related to "fixture modeling" tools (see Section 4.6.2.9, "Fixture modeling and deembedding", on page 212).

- CALCulate:FMODel:DCAL... 971
- CALCulate:FMODel:DEASsistant... 972
- CALCulate:FMODel:DELT... 978
- CALCulate:FMODel:EZD... 989
- CALCulate:FMODel:ISD... 996
- CALCulate:FMODel:SFD... 1007
- CALCulate:FMODel... (other) 1012

CALCulate:FMODel:DCAL...

Commands for the Delta Cal fixture tool.

CALCulate:FMODel:DCAL:APPLy <Channel Id>, <Delta Cal Channel Id>

Uses the calibration of channel <Channel Id> and the Delta Cal calibration of channel <Delta Cal Channel Id> to calculate the S-matrix of the (single-ended) fixture and apply it to deembed the fixture in channel <Channel Id>.

Setting parameters:
<Channel Id> Number of the channel to which the deembedding is applied.

<Delta Cal Channel Id> Number of the Delta Cal channel, i.e. the channel for the on-fixture calibration.

Usage: Setting only

CALCulate:FMODel:DCAL:CREate:CHANnel

Creates a Delta Cal deembedding channel.

Use `CALCulate:FMODel:DCAL:REQuest:CHANnel?` to query its channel number and use it to perform the on-fixture calibration (Delta Cal) in this channel.

Usage: Event

CALCulate:FMODel:DCAL:REQuest:CHANnel?

Usage: Query only

Queries for the number of the Delta Cal deembedding channel created using `CALCulate:FMODel:DCAL:CREate:CHANnel`.

Use the returned channel number to perform the on-fixture calibration (Delta Cal) in this channel.

CALCulate:FMODel:DEASsistant...

These commands implement the functionality of the Deembed Assistant tab and the related dock widget panel.

CALCulate:FMODel:DEASsistant:COUPon:LEFT:CLEar
CALCulate:FMODel:DEASsistant:COUPon:RIGHt:CLEar

Clears the test coupon data available for the lead-in/lead-out.

Usage: Event

Manual operation: See "Coupon A / Coupon B section" on page 734

CALCulate:FMODel:DEASsistant:COUPon:LEFT:FILE <CouponType>, <Path>
CALCulate:FMODel:DEASsistant:COUPon:RIGHt:FILE <CouponType>, <Path>

Loads the properties of a test coupon for the lead-in/lead-out of the DUT.

Parameters:

<CouponType> SYMMetric2x | OPEN1x | SHORt1x

<Path> Path to the Touchstone file, either absolute or relative to the selected working directory (`CALCulate:FMODel:DIRectory`). For `CALCulate:FMODel:DEASsistant:COUPon:<LEFT|RIGHt>:TYPE OPSHort1x`, you have to load a file for OPEN1x **and** SHORt1x (or measure it). Otherwise only for the selected coupon type.

Manual operation: See "Coupon A / Coupon B section" on page 734

CALCulate:FMODel:DEASsistant:COUPon:LEFT:MEASure <CouponType>
CALCulate:FMODel:DEASsistant:COUPon:RIGHt:MEASure <CouponType>

Measures the test coupon.

Before measuring, select the test coupon type using `CALCulate:FMODel:DEASsistant:COUPon:<LEFT|RIGHt>:TYPE` and the ports using `CALCulate:FMODel:DEASsistant:COUPon:<LEFT|RIGHt>:PORT`.

Setting parameters:

<CouponType> SYMMetric2x | OPEN1x | SHORt1x

Test coupon type to be measured
For `CALCulate:FMODel:DEASsistant:COUPon:<LEFT|RIGHt>:TYPE OPSHort1x`, you have to measure OPEN1x **and** SHORt1x (or load it from file). Otherwise measure only the selected coupon type.

Usage: Setting only

Manual operation: See "Coupon A / Coupon B section" on page 734

CALCulate:FMODel:DEASsistant:COUPon:LEFT:PORT <Position>, <PortNum>
CALCulate:FMODel:DEASsistant:COUPon:RIGHt:PORT <Position>, <PortNum>

Defines the port or ports for measuring the selected test coupon (`CALCulate:FMODel:DEASsistant:COUPon:RIGHt:TYPE`)

Parameters:

<Position> UPLeft | UPRight

UPLeft
Left side of the test coupon

UPRight
Right side of the test coupon (`SYMMetric2x` only)

Setting parameters:

<PortNum> Port number (physical port number for single-ended ports, logical port number for balanced ports)

Manual operation: See "Coupon A / Coupon B section" on page 734

CALCulate:FMODel:DEASsistant:COUPon:LEFT:TYPE <CouponType>
CALCulate:FMODel:DEASsistant:COUPon:RIGHt:TYPE <CouponType>

For fixture modeling with the deembedding assistant, this command selects a test coupon to be measured for the lead-in/lead-out, or loaded from file.

The valid test coupons depend on the selected deembedding tool (`CALCulate:FMODel:DEASsistant:TOOL`).

	SYMMetric2x	OPEN1x	SHORt1x	OPSHort1x
ISD	☑	☑	☑	☑
SFD	☑	☑	☑	☐
EZD	☑	☐	☐	☐

- To measure the test coupon, use:
  - `CALCulate:FMODel:DEASsistant:COUPon:<LEFT|RIGHt>:PORT` to select the port(s) at which the test coupon is measured.
  - `CALCulate:FMODel:DEASsistant:COUPon:<LEFT|RIGHt>:MEASure`
- To load the measurement data from file, use `CALCulate:FMODel:DEASsistant:COUPon:<LEFT|RIGHt>:FILE`.

Parameters:

<CouponType> SYMMetric2x | OPEN1x | SHORt1x | OPSHort1x

Test coupon type

SYMMetric2x
Symmetric 2 x Thru

OPEN1x
1 x Open

SHORt1x
1 x Short

Manual operation: See "Coupon A / Coupon B section" on page 734

CALCulate:FMODel:DEASsistant:DUT:CLEar

Clears the data available for DUT + test fixture.

Usage: Event

Manual operation: See "DUT + Fixture section" on page 736

CALCulate:FMODel:DEASsistant:DUT:FILE <Path>

Loads the data of DUT + test fixture from file.

Parameters:

<Path> Path to the Touchstone file, either absolute or relative to the selected working directory (`CALCulate:FMODel:DIRectory`).

Manual operation: See "DUT + Fixture section" on page 736

CALCulate:FMODel:DEASsistant:DUT:MEASure

Measures DUT + fixture at the ports defined using `CALCulate:FMODel:DEASsistant:DUT:PORT`.

Usage: Event

Manual operation: See "DUT + Fixture section" on page 736

CALCulate:FMODel:DEASsistant:DUT:PORT <Position>, <PortNum>

Before you can measure DUT + fixture or run the fixture deembedding tool (`CALCulate:FMODel:DEASsistant:RUN`), you have to define consecutive port ranges for the left/right side of the fixture.

Parameters:

<Position> UPLeft | UPRight | LOLeft | LORight

Defines the port ranges.
For DUTs with only one single-ended or balanced ports on the left/right, specifying an up-port is sufficient for this side.

UPLeft
Upper end of the port range on the left

UPRight
Upper end of the port range on the right

LOLeft
Lower end of the port range on the left

LORight
Lower end of the port range on the right

Manual operation: See "DUT + Fixture section" on page 736

CALCulate:FMODel:DEASsistant:DUT:TYPE <DutType>

Selects the DUT type for the "Deembedding Assistant".

Depending on the number of available test/DUT ports and the selected fixture tool (`CALCulate:FMODel:DEASsistant:TOOL`), choose the appropriate <DutType>.

Parameters:

<DutType> SEX | SEXSe | BX | BXB | BXSE | SE21 | SE22 | NXSE | NXMSe | NXB | NXMB | MULT

SEX
1 x single-ended

SEXSe
1 x 1 single-ended

BX
1 x balanced

BXB
1 x 1 balanced

BXSE
1 x balanced, 1 x single-ended (ISD only)

SE21
2 x 1 single-ended (ISD only)

SE22
2 x 2 single-ended (ISD only)

NXSE
n x single-ended (ISD only)

NXMSe
n x m single-ended (ISD only)

NXB
n x balanced (ISD only)

NXMB
n x m balanced (ISD only)

MULT
Multiport (ISD only)

Manual operation: See "DUT" on page 731

CALCulate:FMODel:DEASsistant:FIXTure:LEFT:TYPE <FixtureType>
CALCulate:FMODel:DEASsistant:FIXTure:RIGHt:TYPE <FixtureType>

Allows you to specify models for the left side (lead-in) and right side (lead-out) of the DUT, depending on the selected DUT type (`CALCulate:FMODel:DEASsistant:DUT:TYPE` on page 975).

Parameters:

<FixtureType> NONE | SE | BALanced | COUPled | UNCoupled | SYMMetrical | MULTiport

NONE
Only for non-existing right sides (lead-outs).

SE
Single-ended; for single-ended lead-ins/outs

BALanced
Balanced; for balanced lead-ins/outs

COUPled
Coupled ports; for lead-ins/outs with more than two logical ports

UNCoupled
Uncoupled ports; for lead-ins/outs with more than two physical ports

SYMMetrical
Symmetrical lead-in/out for balanced DUT ports, consisting of 2 identical single-ended Thru coupons.
Currently only supported for ISD (`CALCulate:FMODel:DEASsistant:TOOL ISD`)

MULTiport
Currently only supported for ISD

Manual operation: See "Left: Model A / Right: Model A|B" on page 731

CALCulate:FMODel:DEASsistant:IMPCorrect <Boolean>

Activates impedance correction during DUT + test fixture measurement.

Parameters:

<Boolean>
ON | 1
Impedance correction activated
OFF | 0
Impedance correction deactivated

Manual operation: See "DUT + Fixture section" on page 736

CALCulate:FMODel:DEASsistant:PRESet

Resets the DUT and fixture settings of the deembedding assistant to their respective defaults.

Note that a `*RST` does not affect these settings.

Usage: Event

Manual operation: See "DUT" on page 731

CALCulate:FMODel:DEASsistant:RUN

Runs the selected deembedding (`CALCulate:FMODel:DEASsistant:TOOL`) tool and applies the calculated fixture deembedding to the ports selected using `CALCulate:FMODel:DEASsistant:DUT:PORT`.

Usage: Event

Manual operation: See "DUT + Fixture section" on page 736

CALCulate:FMODel:DEASsistant:RUN:RESult?

Returns whether the latest deembedding tool execution using `CALCulate:FMODel:DEASsistant:RUN` was successful (PASS) or not (FAIL), or if it has not been run before (NOT_RUN).

Usage: Query only

CALCulate:FMODel:DEASsistant:SAMCoupon <Boolean>

For the deembedding assistant, this command defines whether only the left coupon model is measured or loaded from file.

This simplification is allowed for two-sided DUTs using the same coupon model for the left and right side (`CALCulate:FMODel:DEASsistant:COUPon:LEFT|RIGHt:TYPE`).

Parameters:

<Boolean> ON (1) | OFF (0), where ON means only the left coupon model is measured (`CALCulate:FMODel:DEASsistant:COUPon:LEFT:MEASure`) or loaded from file (`CALCulate:FMODel:DEASsistant:COUPon:LEFT:FILE`).

Manual operation: See "Use Same Coupon Left and Right" on page 733

CALCulate:FMODel:DEASsistant:TOOL <FixtureTool>

Selects the fixture modeling tool to be used in the deembedding assistant.

Parameters:

<FixtureTool> ISD | SFD | EZD | DCAL

ISD
AtaiTec's *In Situ De-Embedding* tool
See Section 4.7.10, "In-situ de-embedding", on page 255

SFD
PacketMicro's *Smart Fixture De-embedding* tool
See Section 4.7.11, "Smart fixture de-embedding", on page 255

EZD
Eazy Deembedding based on IEEE 370
See Section 4.7.9, "Eazy de-embedding based on IEEE 370", on page 253

DCAL
Delta Cal

Manual operation: See "Fixture Tool/Run Tool" on page 748

CALCulate:FMODel:DELT...

Commands for *Delta-L* PCB characterization (see Section 4.7.12, "Delta-L 4.0 PCB characterization", on page 256).

CALCulate:FMODel:DELT:CUToff:FREQuency 979
CALCulate:FMODel:DELT:CUToff:STATe 979
CALCulate:FMODel:DELT:DIRectory 980
CALCulate:FMODel:DELT:DIRectory:DEFault 980
CALCulate:FMODel:DELT:FITTing:ORDer 980
CALCulate:FMODel:DELT:FREQuencies:CURRent? 980
CALCulate:FMODel:DELT:FREQuencies:CURRent:COUNt? 981
CALCulate:FMODel:DELT:FREQuencies:DEFault? 981
CALCulate:FMODel:DELT:FREQuencies:DEFault:COUNt? 981
CALCulate:FMODel:DELT:FREQuencies:USEDefault 981
CALCulate:FMODel:DELT:FREQuencies:USER? 982
CALCulate:FMODel:DELT:FREQuencies:USER:ADD 982
CALCulate:FMODel:DELT:FREQuencies:USER:DELete 982
CALCulate:FMODel:DELT:FREQuencies:USER:DELete:ALL 983
CALCulate:FMODel:DELT:FREQuencies:USER:COUNt? 983
CALCulate:FMODel:DELT:M1L:CACHe:CLEar:ALL 983

CALCulate:FMODel:DELT:M1L:CACHe:CLEar:SELected....983
CALCulate:FMODel:DELT:M1L:DIFFmode....984
CALCulate:FMODel:DELT:M1L:FILename....984
CALCulate:FMODel:DELT:M2L:FILename....984
CALCulate:FMODel:DELT:M3L:FILename....984
CALCulate:FMODel:DELT:M1L:LENGth....984
CALCulate:FMODel:DELT:M2L:LENGth....984
CALCulate:FMODel:DELT:M3L:LENGth....984
CALCulate:FMODel:DELT:M1L:MEASure....985
CALCulate:FMODel:DELT:M2L:MEASure....985
CALCulate:FMODel:DELT:M3L:MEASure....985
CALCulate:FMODel:DELT<Ph_pt>:M1L[:STATe]....985
CALCulate:FMODel:DELT<Ph_pt>:M2L[:STATe]....985
CALCulate:FMODel:DELT<Ph_pt>:M3L[:STATe]....985
CALCulate:FMODel:DELT:MEASurement....985
CALCulate:FMODel:DELT:METHod....986
CALCulate:FMODel:DELT:PORDer....986
CALCulate:FMODel:DELT:RESonance....986
CALCulate:FMODel:DELT:RUN....987
CALCulate:FMODel:DELT:SWEep:CONTrol....987
CALCulate:FMODel:DELT:SWEep:CONTrol:IFBW?....987
CALCulate:FMODel:DELT:SWEep:CONTrol:STEP?....987
CALCulate:FMODel:DELT:SWEep:FREQuency:MAXimum....988
CALCulate:FMODel:DELT:SWEep:FREQuency:MINimum....988
CALCulate:FMODel:DELT:TCONfig....988
CALCulate:FMODel:DELT:TDR....988

CALCulate:FMODel:DELT:CUToff:FREQuency <Frequency>

If applying a cutoff frequency is enabled (`CALCulate:FMODel:DELT:CUToff:STATe ON`, this command defines the cutoff frequency.

Parameters:

<Frequency> Cutoff frequency

*RST: 25 GHz
Default unit: Hz

Options: R&S ZNB-K231

Manual operation: See "Options" on page 773

CALCulate:FMODel:DELT:CUToff:STATe <Boolean>

For AITT version 2024.05 or higher, this command allows you to apply a cutoff frequency or not.

Specify the cutoff frequency using `CALCulate:FMODel:DELT:CUToff:FREQuency`.

For earlier AITT versions, the cutoff frequeny is always applied.

Parameters:

<Boolean>

ON (1)
Enable cutoff frequency

OFF (0)
Disable cutoff frequency

*RST: OFF

Options: R&S ZNB-K231

Manual operation: See "Options" on page 773

CALCulate:FMODel:DELT:DIRectory <String>

Defines the working directory of the Delta-L tool.

Parameters:

<String> Directory path, either absolute or relative to the current directory (`MMEMory:CDIRectory`).

Options: R&S ZNB-K231

Manual operation: See "Output Settings" on page 774

CALCulate:FMODel:DELT:DIRectory:DEFault

Resets the working directory of the Delta-L tool to its default `C:\Users\Public\Documents\Rohde-Schwarz\Vna\DeltaL`.

Usage: Event

Options: R&S ZNB-K231

Manual operation: See "Output Settings" on page 774

CALCulate:FMODel:DELT:FITTing:ORDer <FittingOrder>

Defines the fitting weight order (AITT version 2024.05 or higher).

Parameters:

<FittingOrder> FWO0 | FWO1 | FWO2 | FWO3

Orders 0 to 4

*RST: 3

Options: R&S ZNB-K231

Manual operation: See "Options" on page 773

CALCulate:FMODel:DELT:FREQuencies:CURRent?

Returns the frequencies at which the Delta-L algorithm calculates the loss-per-inch uncertainties.

You can use the algorithm's default or custom frequencies (see `CALCulate:FMODel:DELT:FREQuencies:USEDefault`).

Usage: Query only

Options: R&S ZNB-K231

Manual operation: See "Frequencies" on page 773

CALCulate:FMODel:DELT:FREQuencies:CURRent:COUNt?

Returns at how many frequencies the Delta-L algorithm calculates the loss-per-inch uncertainties.

Use `CALCulate:FMODel:DELT:FREQuencies:CURRent?` to query the list of frequencies.

Usage: Query only

Options: R&S ZNB-K231

Manual operation: See "Frequencies" on page 773

CALCulate:FMODel:DELT:FREQuencies:DEFault?

Returns the frequencies at which the Delta-L algorithm calculates the loss-per-inch uncertainties per default.

You can either use the algorithm's default or custom frequencies (see `CALCulate:FMODel:DELT:FREQuencies:USEDefault`).

Usage: Query only

Options: R&S ZNB-K231

Manual operation: See "Frequencies" on page 773

CALCulate:FMODel:DELT:FREQuencies:DEFault:COUNt?

Returns the number of frequencies at which the Delta-L algorithm calculates the loss-per-inch uncertainties per default.

Use `CALCulate:FMODel:DELT:FREQuencies:DEFault?` to query the frequencies.

Usage: Query only

Options: R&S ZNB-K231

Manual operation: See "Frequencies" on page 773

CALCulate:FMODel:DELT:FREQuencies:USEDefault <Boolean>

Delta-L computes the uncertainty in the loss-per-inch calculation at specified frequencies.

This command specifies whether to use the tool's default frequencies or custom frequencies, which can be defined using `CALCulate:FMODel:DELT<Ph_pt>:FREQuencies:USER` commands.

Parameters:

<Boolean>

ON (1)
Use default frequencies

OFF (0)
Use custom frequencies

Options: R&S ZNB-K231

Manual operation: See "Frequencies" on page 773

CALCulate:FMODel:DELT:FREQuencies:USER?

The Delta-L algorithm can calculate the loss-per-inch uncertainties either at default or at custom frequencies (see `CALCulate:FMODel:DELT:FREQuencies:USEDefault`).

This command returns the custom frequencies, defined using `CALCulate:FMODel:DELT:FREQuencies:USER:ADD` etc.

Usage: Query only

Options: R&S ZNB-K231

Manual operation: See "Frequencies" on page 773

CALCulate:FMODel:DELT:FREQuencies:USER:ADD <UserFrequencies>
CALCulate:FMODel:DELT:FREQuencies:USER:DELete <Frequency>

The Delta-L algorithm can calculate the loss-per-inch uncertainties either at default or at custom frequencies (see `CALCulate:FMODel:DELT:FREQuencies:USEDefault`).

`CALCulate:FMODel:DELT<Ph_pt>:FREQuencies:USER:ADD|DELete` adds/deletes a frequency to|from the list of custom frequencies.

Setting parameters:

<Frequency>

Custom frequency

Range: The frequency range of the analyzer
Default unit: Hz

Usage: Setting only

Options: R&S ZNB-K231

Manual operation: See "Frequencies" on page 773

CALCulate:FMODel:DELT:FREQuencies:USER:DELete:ALL

The Delta-L algorithm can calculate the loss-per-inch uncertainties either at default or at custom frequencies (see `CALCulate:FMODel:DELT:FREQuencies:USEDefault`).

This command clears the list of custom frequencies.

Usage: Event

Options: R&S ZNB-K231

Manual operation: See "Frequencies" on page 773

CALCulate:FMODel:DELT:FREQuencies:USER:COUNt?

The Delta-L algorithm can calculate the loss-per-inch uncertainties either at default or at custom frequencies (see `CALCulate:FMODel:DELT:FREQuencies:USEDefault`).

This command returns the number of custom frequencies, defined using `CALCulate:FMODel:DELT:FREQuencies:USER:ADD` etc.

Usage: Query only

Options: R&S ZNB-K231

Manual operation: See "Frequencies" on page 773

CALCulate:FMODel:DELT:M1L:CACHe:CLEar:ALL

If the 1-length method is used (`CALCulate:FMODel:DELT:METHod M1L`), the VNA firmware builds up a cache of coupon measurements (Touchstone files) in the `DeltaL\1LMeasurementCache` subfolder of the user data folder `C:\Users\Public\Documents\Rohde-Schwarz\Vna`.

This command clears this cache.

Usage: Event

Options: R&S ZNB-K231

Manual operation: See "Clear All 1L Measurements/Clear Selected 1L Measurements" on page 776

CALCulate:FMODel:DELT:M1L:CACHe:CLEar:SELected <String>

If the 1-length method is used (`CALCulate:FMODel:DELT:METHod M1L`), the VNA firmware builds up a cache of coupon measurements (Touchstone files) in the `DeltaL\1LMeasurementCache` subfolder of the user data folder `C:\Users\Public\Documents\Rohde-Schwarz\Vna`.

This command deletes the selected file from this cache.

Setting parameters:
<String> Path to the Touchstone file, relative to the cache directory.

Usage: Setting only

Options: R&S ZNB-K231

Manual operation: See "Clear All 1L Measurements/Clear Selected 1L Measurements" on page 776

CALCulate:FMODel:DELT:M1L:DIFFmode <Boolean>

The one-length method (`CALCulate:FMODel:DELT:METHod M1L`) handles single-ended and balanced measurement results differently. This command selects between the two modes.

Parameters:
<Boolean>
OFF (0)
Single-ended mode
ON (1)
Differential mode
*RST: OFF (0)

Options: R&S ZNB-K231

Manual operation: See "Method" on page 772

CALCulate:FMODel:DELT:M1L:FILename <String>
CALCulate:FMODel:DELT:M2L:FILename <String>
CALCulate:FMODel:DELT:M3L:FILename <String>

For a Delta-L+ PCB characterization, this command loads the "1L Measurement" data from a Touchstone file.

Parameters:
<String> Path to the Touchstone file, either absolute or relative to the selected working directory (`CALCulate:FMODel:DELT:DIRectory`).

Options: R&S ZNB-K231

Manual operation: See "Load File" on page 776

CALCulate:FMODel:DELT:M1L:LENGth <Length>
CALCulate:FMODel:DELT:M2L:LENGth <Length>
CALCulate:FMODel:DELT:M3L:LENGth <Length>

Specifies the length of the 1st/2nd/3rd line in Delta-L measurements (`CALCulate:FMODel:DELT<Ph_pt>:M1L|M2L|M3L:MEASure`).

Parameters:
<Length> Length in inches

Options: R&S ZNB-K231

Manual operation: See "Length" on page 776

CALCulate:FMODel:DELT:M1L:MEASure
CALCulate:FMODel:DELT:M2L:MEASure
CALCulate:FMODel:DELT:M3L:MEASure

Measures the full set of S-parameters at the active ports (see `CALCulate:FMODel:DELT<Ph_pt>:M1L|M2L|M3L[:STATe]`).

Usage: Event

Options: R&S ZNB-K231

Manual operation: See "Measure" on page 776

CALCulate:FMODel:DELT<Ph_pt>:M1L[:STATe] <Boolean>
CALCulate:FMODel:DELT<Ph_pt>:M2L[:STATe] <Boolean>
CALCulate:FMODel:DELT<Ph_pt>:M3L[:STATe] <Boolean>

Defines the active ports for the 1st/2nd/3rd line Delta-L measurement (`CALCulate:FMODel:DELT<Ph_pt>:M1L|M2L|M3L:MEASure`).

For each measurement, either two or four ports must be active. If the 1-length method is used (`CALCulate:FMODel:DELT:METHod M1L`), then four ports must be active for the first line.

Suffix:
<Ph_pt> Physical port number

Parameters:
<Boolean>
ON (1)
Port <Ph_pt> is active in the respective measurement.
OFF (0)
Port <Ph_pt> is inactive in the respective measurement.

Options: R&S ZNB-K231

Manual operation: See "Ports table" on page 776

CALCulate:FMODel:DELT:MEASurement <Measurement>

Defines whether single-ended or balanced ports are measured for Delta-L characterizations.

Parameters:
<Measurement> SINGleended | BALanced
SINGleended
Single-ended ports only
BALanced
Balanced ports only

*RST: SINGleended

Options: R&S ZNB-K231

Manual operation: See "Measurements" on page 772

CALCulate:FMODel:DELT:METHod <Method>

Selects the Delta-L+ characterization method to be used.

Parameters:

<Method> M1L | M2L | M3L

One-, two-, or three-line/length analysis

Options: R&S ZNB-K231

Manual operation: See "Method" on page 772

CALCulate:FMODel:DELT:PORDer <PortOrder>

Declares how the DUT is connected to the measurement system. Choosing the wrong port ordering for your setup, can result in erroneous results.

Parameters:

<PortOrder> ODDeven | SEQuential

ODDeven
Ports 1, 3 and 2, 4 form input/output pairs

SEQuential
Ports 1, 2 and 3, 4 form input/output pairs

Options: R&S ZNB-K231

Manual operation: See "Port Order" on page 773

CALCulate:FMODel:DELT:RESonance <Boolean>

Enables or disables the improved Delta-L algorithms that implement resonance removal.

Parameters:

<Boolean> **ON (1)**
Enable resonance removal

OFF (0)
Disable resonance removal

*RST: OFF

Options: R&S ZNB-K231

Manual operation: See "Options" on page 773

CALCulate:FMODel:DELT:RUN

Runs the AITT-DL tool with the collected measurement data and the configured Delta-L settings.

Usage: Event

Options: R&S ZNB-K231

Manual operation: See "Run" on page 776

CALCulate:FMODel:DELT:SWEep:CONTrol <SweepControl>

Tells the firmware to use either default values for frequency step size and IF bandwidth, or the settings of the active channel for the Delta-L measurement.

The resulting frequency step size and IF bandwidth can be queried using `CALCulate:FMODel:DELT:SWEep:CONTrol:STEP?` on page 987 and `CALCulate:FMODel:DELT:SWEep:CONTrol:IFBW?` on page 987, respectively.

Parameters:

<SweepControl> DEFault | CURRent

DEFault
Use the default values of the Delta-L algorithm:
10 MHz frequency step size, 1 kHz IF bandwidth

SweepControl
Use the values of the active channel.

Options: R&S ZNB-K231

Manual operation: See "Sweep Control" on page 774

CALCulate:FMODel:DELT:SWEep:CONTrol:IFBW?

For the Delta-L measurement, the firmware can either use a default IF bandwidth, or the active channel setting (see `CALCulate:FMODel:DELT:SWEep:CONTrol`).

This query returns the resulting IF bandwidth.

Usage: Query only

Options: R&S ZNB-K231

Manual operation: See "Sweep Control" on page 774

CALCulate:FMODel:DELT:SWEep:CONTrol:STEP?

For the Delta-L measurement, the firmware can either use a default frequency step size, or the active channel setting (see `CALCulate:FMODel:DELT:SWEep:CONTrol`).

This query returns the resulting frequency step size.

Usage: Query only

Options: R&S ZNB-K231

Manual operation: See "Sweep Control" on page 774

CALCulate:FMODel:DELT:SWEep:FREQuency:MAXimum <Frequency>
CALCulate:FMODel:DELT:SWEep:FREQuency:MINimum <Frequency>

Defines the sweep range of the Delta-L measurement channel. The entered MAXimum and MINimum values are set as `[SENSe<Ch>:]FREQuency:STOP` and `[SENSe<Ch>:]FREQuency:STARt`, respectively.

Furthermore, the VNA collects data for the LPI computation a the MAXimum and MINimum frequencies (in addition to `CALCulate:FMODel:DELT:FREQuencies:CURRent?`).

Parameters:

<Frequency> Frequency values
The default MINimum frequency is the standard Delta-L start frequency of 10 MHz. The default MAXimum frequency is either 20 GHz or the instrument's maximum frequency, whichever is smaller.
Default unit: Hz

Options: R&S ZNB-K231

Manual operation: See "Sweep Frequency" on page 774

CALCulate:FMODel:DELT:TCONfig <Boolean>

Defines how the analyzer firmware sets up the diagram area for the Delta-L measurement.

Parameters:

<Boolean> **ON (1)**
Use the recommended trace configuration, i.e. display each of the traces in a separate diagram.

OFF (0)
Display all traces in a single diagram.

Options: R&S ZNB-K231

Manual operation: See "Trace" on page 774

CALCulate:FMODel:DELT:TDR <Boolean>

If the Time domain analysis option R&S ZNB-K2 is installed, this command allows you to add the TDR-based converted impedance traces to the recommended trace configuration.

Parameters:

<Boolean> **OFF (0)**
TDR-based impedance traces added

ON (1)
TDR-based impedance traces not added

*RST: OFF

Options: R&S ZNB-K231

Manual operation: See "Trace" on page 774

CALCulate:FMODel:EZD...

Commands for *Eazy Deembedding* (see Section 4.7.9, "Eazy de-embedding based on IEEE 370", on page 253).

CALCulate:FMODel:EZD<Ph_pt>:CAUSality 989
CALCulate:FMODel:EZD:COUPon:MEASure 990
CALCulate:FMODel:EZD:COUPon:MEASure:FILename 990
CALCulate:FMODel:EZD:COUPon:MEASure:FILename:CLEar 990
CALCulate:FMODel:EZD<Ph_pt>:COUPon[:STATe] 990
CALCulate:FMODel:EZD:DCEXtrapolat 991
CALCulate:FMODel:EZD<Ph_pt>:DELay:LEFT 991
CALCulate:FMODel:EZD<Ph_pt>:DELay:RIGHt 991
CALCulate:FMODel:EZD:DEPLane<Pl> 991
CALCulate:FMODel:EZD:DEPLane:STATe 991
CALCulate:FMODel:EZD:DUT:MEASure 992
CALCulate:FMODel:EZD:DUT:MEASure:FILename 992
CALCulate:FMODel:EZD:DUT:MEASure:FILename:CLEar 992
CALCulate:FMODel:EZD<Ph_pt>:DUT[:STATe] 992
CALCulate:FMODel:EZD<Ph_pt>:FASTmode 993
CALCulate:FMODel:EZD:GENerate:SIDE<1|2> 993
CALCulate:FMODel:EZD<Ph_pt>:HFC 993
CALCulate:FMODel:EZD<Ph_pt>:ICALculation 993
CALCulate:FMODel:EZD:IMPedance 994
CALCulate:FMODel:EZD:IMPedance:BWIDlimit 994
CALCulate:FMODel:EZD:IMPedance:REFerence 994
CALCulate:FMODel:EZD:PORT:ORDer 994
CALCulate:FMODel:EZD:PRESet 995
CALCulate:FMODel:EZD:RUN:RUN 995
CALCulate:FMODel:EZD<Pt>:RUN[:STATe] 995
CALCulate:FMODel:EZD<Ph_pt>:SURFace 996
CALCulate:FMODel:EZD:VERSion? 996

CALCulate:FMODel:EZD<Ph_pt>:CAUSality <Causality>

Suffix:
<Ph_pt>

Parameters:
<Causality>

Options: R&S ZNB-K210

Manual operation: See "Causality Correction" on page 728

CALCulate:FMODel:EZD:COUPon:MEASure

For a fixture modeling with the Easy De-embedding (EZD) tool, this command starts the measurement of the symmetrical 2xThru coupon at the active ports (see `CALCulate:FMODel:EZD<Ph_pt>:COUPon[:STATe]` on page 990).

Usage: Event

Options: R&S ZNB-K210

Manual operation: See "Measure" on page 717

CALCulate:FMODel:EZD:COUPon:MEASure:FILename <String>

For a fixture modeling with the EZD tool, this command loads the coupon properties from a Touchstone file.

Parameters:

<String> Path to the Touchstone file, either absolute or relative to the selected working directory (`CALCulate:FMODel:DIRectory`).

Options: R&S ZNB-K210

Manual operation: See "Load File" on page 718

CALCulate:FMODel:EZD:COUPon:MEASure:FILename:CLEar

Invalidates the file path previously set using `CALCulate:FMODel:EZD:COUPon:MEASure:FILename`.

Usage: Event

Options: R&S ZNB-K210

CALCulate:FMODel:EZD<Ph_pt>:COUPon[:STATe] <Boolean>

For a fixture modeling with the EZD tool, this command allows you to specify the ports to which the test coupon is connected.

This must be defined before measuring the test coupon (using `CALCulate:FMODel:EZD:COUPon:MEASure`).

Suffix:

<Ph_pt> Physical port number

Parameters:

<Boolean>

ON (1)
If the test coupon is connected to port <Ph_pt>

OFF (0)
Otherwise

Options: R&S ZNB-K210

Manual operation: See "Active" on page 718

CALCulate:FMODel:EZD:DCEXtrapolat <Boolean>

Defines whether the Touchstone de-embedding file generated by the EZD tool contains data for DC.

Parameters:

<Boolean>
ON (1)
The EZD tool adds (extrapolated) S-parameter data for f = 0.
OFF (0)
The frequencies only reach down to the start frequency of the instrument sweep.

Options: R&S ZNB-K210

Manual operation: See "DC Extrapolation" on page 726

CALCulate:FMODel:EZD<Ph_pt>:DELay:LEFT <Value>
CALCulate:FMODel:EZD<Ph_pt>:DELay:RIGHt <Value>

Suffix:
<Ph_pt>

Parameters:

<Value> Default unit: s

Options: R&S ZNB-K210

Manual operation: See "Flight Time Scaling" on page 728

CALCulate:FMODel:EZD:DEPLane<Pl> <Value>

Selects the position of the deembedding plane in the TDR impedance profile traces for side 1 (left) and 2 (right).

Only takes effect if `CALCulate:FMODel:EZD:DEPLane:STATe` is ON.

This is a global setting.

Suffix:

<Pl> 1 = left, 2 = right

Parameters:

<Value> Default unit: s

Options: R&S ZNB-K210

Manual operation: See "Deembedding Plane" on page 726

CALCulate:FMODel:EZD:DEPLane:STATe <Boolean>

Enables or disables deembedding plane selection in the deembedding assistant.

Use `CALCulate:FMODel:EZD:DEPLane<Pl>` on page 991 to set the position in the TDR impedance profile traces.

This is a global setting.

Parameters:
<Boolean>

Options: R&S ZNB-K210

Manual operation: See "Deembedding Plane" on page 726

CALCulate:FMODel:EZD:DUT:MEASure

For a fixture modeling with the EZD tool, this command allows you to measure "DUT + Test Fixture" at the active ports (see `CALCulate:FMODel:EZD<Ph_pt>:DUT[:STATe]`)

Usage: Event

Options: R&S ZNB-K210

Manual operation: See "Measure" on page 719

CALCulate:FMODel:EZD:DUT:MEASure:FILename <String>

For a fixture modeling with the EZD tool, this command loads the "DUT + Test Fixture" properties from a Touchstone file.

Parameters:
<String> Path to the Touchstone file, either absolute or relative to the selected working directory (`CALCulate:FMODel:DIRectory`).

Options: R&S ZNB-K210

Manual operation: See "Load File" on page 719

CALCulate:FMODel:EZD:DUT:MEASure:FILename:CLEar

Invalidates the file path previously set using `CALCulate:FMODel:EZD:DUT:MEASure:FILename`.

Usage: Event

Options: R&S ZNB-K210

Manual operation: See "Load File" on page 718

CALCulate:FMODel:EZD<Ph_pt>:DUT[:STATe] <Boolean>

Defines the physical ports to which DUT + test fixture are connected.

Suffix:
<Ph_pt> Physical port number

Parameters:
<Boolean> **ON (1)**
Port <Ph_pt> connected

OFF (0)
Port <Ph_pt> not connected

Options: R&S ZNB-K210

Manual operation: See "Active" on page 719

CALCulate:FMODel:EZD<Ph_pt>:FASTmode <Boolean>

Suffix:
<Ph_pt>

Parameters:
<Boolean>

Options: R&S ZNB-K210

Manual operation: See "Speed Adjustment" on page 727

CALCulate:FMODel:EZD:GENerate:SIDE<1|2> <Boolean>

Tells the EZD tool to create fixture deembedding data at side 1, at side 2, or at both sides.

Suffix:
<1|2> Side

Parameters:
<Boolean> **ON (1)**
Create fixture deembedding data at the specified side (<1|2>).

OFF (0)
Do not create fixture deembedding data at the specified side (<1|2>).

Options: R&S ZNB-K210

CALCulate:FMODel:EZD<Ph_pt>:HFC <Boolean>

Suffix:
<Ph_pt>

Parameters:
<Boolean>

Options: R&S ZNB-K210

Manual operation: See "High Frequency Correction" on page 729

CALCulate:FMODel:EZD<Ph_pt>:ICALculation <CalcType>

Suffix:
<Ph_pt>

Parameters:
<CalcType> FITTed_rlgc | DIRect_rlgc | CONStant

Options: R&S ZNB-K210

Manual operation: See "Impedance Calculation" on page 728

CALCulate:FMODel:EZD:IMPedance <Boolean>

Switches the impedance correction of the EZD tool ON or OFF. Its reference impedance and bandwidth limit parameters can be set using `CALCulate:FMODel:EZD:IMPedance:REFerence` and `CALCulate:FMODel:EZD:IMPedance:BWIDlimit`, respectively.

This is a global setting.

Parameters:
<Boolean> ON (1) or OFF (0)

Options: R&S ZNB-K210

Manual operation: See "Impedance Correction" on page 716

CALCulate:FMODel:EZD:IMPedance:BWIDlimit <Value>

Defines the "Bandwidth Limit" parameter of the EZD tool's impedance correction (see `CALCulate:FMODel:EZD:IMPedance`.

Parameters:
<Value> Default unit: Hz

Options: R&S ZNB-K210

Manual operation: See "High Frequency Correction" on page 729

CALCulate:FMODel:EZD:IMPedance:REFerence <Value>

Defines the reference impedance for the impedance correction of the EZD tool (see `CALCulate:FMODel:EZD:IMPedance`).

Parameters:
<Value> Default unit: Ohm

Options: R&S ZNB-K210

CALCulate:FMODel:EZD:PORT:ORDer <PortOrder>

Tells the EZD tool about the port ordering of the test fixture.

Parameters:
<PortOrder> ODD | NON

ODD
Odd ports are on the left and even ports are on the right.

NON
Ports 1 to N are on the left and ports N+1 to 2·N are on the right.

Options: R&S ZNB-K210

Manual operation: See "Port Order" on page 726

CALCulate:FMODel:EZD:PRESet

Restores the global default settings of the EZD tool.

Usage: Event

Options: R&S ZNB-K210

Manual operation: See "Reset to Default" on page 716

CALCulate:FMODel:EZD:RUN:RUN

Runs the EZD tool.

Before executing this command, make sure that:

- The test coupon measurement (using `CALCulate:FMODel:EZD:COUPon:MEASure`) finished successfully, or the test coupon data were successfully loaded from file (using `CALCulate:FMODel:EZD:COUPon:MEASure:FILename`
- The measurement of DUT + test fixture (using `CALCulate:FMODel:EZD:COUPon:MEASure`) finished successfully.

The resulting Touchstone files are written to C:\Users\Public\Documents\Rohde-Schwarz\Vna\Embedding. If result files with the same name already exist, they are overwritten.

Usage: Event

Options: R&S ZNB-K210

Manual operation: See "Run <Fixture Modeling Tool>" on page 719

CALCulate:FMODel:EZD<Pt>:RUN[:STATe] <Boolean>

Tells the analyzer to assign deembedding files to certain ports, after the EZD tool has run.

Parameters:

<Boolean>

ON (1)
Assign a deembedding file to port <Pt>.

OFF (0)
Otherwise

Options: R&S ZNB-K210

Manual operation: See "Apply" on page 720

CALCulate:FMODel:EZD<Ph_pt>:SURFace <SurfaceRoughness>

Suffix:
<Ph_pt>

Parameters:
<SurfaceRoughness>

Options: R&S ZNB-K210

Manual operation: See "Surface Roughness RMS" on page 729

CALCulate:FMODel:EZD:VERSion?

Queries the installed version of the EZD tool.

Usage: Query only

Options: R&S ZNB-K210

Manual operation: See "Version" on page 730

CALCulate:FMODel:ISD...

Commands for AtaiTec's *In Situ De-Embedding* (ISD) tool, see http://ataitec.com/products/isd/.

CALCulate:FMODel:ISD:ATTenuation:BEHavior....997
CALCulate:FMODel:ISD:COUPon:MEASure....997
CALCulate:FMODel:ISD:COUPon:MEASure:FILename....998
CALCulate:FMODel:ISD:COUPon:MEASure:FILename:CLEar....998
CALCulate:FMODel:ISD:COUPon:MEASure:OPEN....998
CALCulate:FMODel:ISD:COUPon:MEASure:OPEN:FILename....998
CALCulate:FMODel:ISD:COUPon:MEASure:SHORt....999
CALCulate:FMODel:ISD:COUPon:MEASure:SHORt:FILename....999
CALCulate:FMODel:ISD<Ph_pt>:COUPon[:STATe]....999
CALCulate:FMODel:ISD:COUPon:TYPE....999
CALCulate:FMODel:ISD:DCEXtrapolat....1000
CALCulate:FMODel:ISD:DIFFonly....1000
CALCulate:FMODel:ISD:DUT:MEASure....1000
CALCulate:FMODel:ISD:DUT:MEASure:FILename....1001
CALCulate:FMODel:ISD:DUT:MEASure:FILename:CLEar....1001
CALCulate:FMODel:ISD<Ph_pt>:DUT[:STATe]....1001
CALCulate:FMODel:ISD:DUT:TYPE....1001
CALCulate:FMODel:ISD:FTIMe:DUT....1002
CALCulate:FMODel:ISD:FTIMe:OVERride....1002
CALCulate:FMODel:ISD:IMPedance....1002
CALCulate:FMODel:ISD:OPERation....1003
CALCulate:FMODel:ISD:PASSivity....1003
CALCulate:FMODel:ISD:PORT:ORDer....1003
CALCulate:FMODel:ISD:PORT:SKIP....1004
CALCulate:FMODel:ISD:PORT:SKIP:LEFT....1004
CALCulate:FMODel:ISD:PORT:SKIP:NONE....1004

CALCulate:FMODel:ISD:PORT:SKIP:RIGHt....1004
CALCulate:FMODel:ISD:PRESet....1004
CALCulate:FMODel:ISD:RUN:RUN....1004
CALCulate:FMODel:ISD<Pt>:RUN[:STATe]....1005
CALCulate:FMODel:ISD:SCALe:ATTenuation....1005
CALCulate:FMODel:ISD:SCALe:FREQuency....1005
CALCulate:FMODel:ISD:SCALe:FTIMe....1006
CALCulate:FMODel:ISD:SCALe:FTIMe:TYPE....1006
CALCulate:FMODel:ISD:SMALIfixture....1006
CALCulate:FMODel:ISD:SMOThing....1007
CALCulate:FMODel:ISD:TRACe:COUPling....1007
CALCulate:FMODel:ISD:VERSion?....1007

CALCulate:FMODel:ISD:ATTenuation:BEHavior <AttenuationBehavior>

Sets/gets the `linear_2x` batch mode parameter of the ISD tool

Parameters:
<AttenuationBehavior>RESonant | LINear | NONLinear

LINear
Used for linear insertion loss of test coupon (parameter value `1`)

NONLinear
Used for nonlinear insertion loss of test coupon (parameter value `2`)

RESonant
The 2x Thru test coupon is split and used directly for deembedding (parameter value `3`).
This option can be more accurate when the fixture and 2x Thru have the same impedance at every location.

Options: R&S ZNB-K220

Manual operation: See "Test Coupons > Insertion Loss" on page 722

CALCulate:FMODel:ISD:COUPon:MEASure

For a fixture modeling with the ISD tool and coupon types SYMMetric2x | OPEN1x | SHORt1x (see `CALCulate:FMODel:ISD:COUPon:TYPE`), this command starts the measurement of the coupon at the active ports (see `CALCulate:FMODel:ISD<Ph_pt>:COUPon[:STATe]`).

Example: See Section 8.2.8, "Fixture modeling", on page 1685

Usage: Event

Options: R&S ZNB-K220

Manual operation: See "Measure" on page 717

CALCulate:FMODel:ISD:COUPon:MEASure:FILename <String>

For a fixture modeling with the ISD tool and coupon types SYMMetric2x | OPEN1x | SHORt1x (see `CALCulate:FMODel:ISD:COUPon:TYPE`), this command loads the coupon properties from a Touchstone file.

Parameters:

<String> Path to the Touchstone file, either absolute or relative to the selected working directory (`CALCulate:FMODel:DIRectory`).

Options: R&S ZNB-K220

Manual operation: See "Load File" on page 718

CALCulate:FMODel:ISD:COUPon:MEASure:FILename:CLEar

Invalidates the file path previously set using `CALCulate:FMODel:ISD:DUT:MEASure:FILename`, `CALCulate:FMODel:ISD:COUPon:MEASure:OPEN:FILename`, or `CALCulate:FMODel:ISD:COUPon:MEASure:SHORt:FILename`.

Usage: Event

Options: R&S ZNB-K220

Manual operation: See "Load File" on page 718

CALCulate:FMODel:ISD:COUPon:MEASure:OPEN

For a fixture modeling with the ISD tool and coupon type OPSHort1x (see `CALCulate:FMODel:ISD:COUPon:TYPE`), this command starts the measurement of the **Open** coupon at the active ports (see `CALCulate:FMODel:ISD<Ph_pt>:COUPon[:STATe]`).

Usage: Event

Options: R&S ZNB-K220

Manual operation: See "Measure" on page 717

CALCulate:FMODel:ISD:COUPon:MEASure:OPEN:FILename <String>

For a fixture modeling with the ISD tool and coupon type OPSHort1x (see `CALCulate:FMODel:ISD:COUPon:TYPE`), this command loads the properties of the Open coupon from a Touchstone file.

Parameters:

<String> Path to the Touchstone file, either absolute or relative to the selected working directory (`CALCulate:FMODel:DIRectory`).

Options: R&S ZNB-K220

Manual operation: See "Load File" on page 718

CALCulate:FMODel:ISD:COUPon:MEASure:SHORt

For a fixture modeling with the ISD tool and coupon type OPSHort1x (see `CALCulate:FMODel:ISD:COUPon:TYPE`), this command starts the measurement of the **Short** coupon at the active ports (see `CALCulate:FMODel:ISD<Ph_pt>:COUPon[:STATe]`).

Usage: Event

Options: R&S ZNB-K220

Manual operation: See "Measure" on page 717

CALCulate:FMODel:ISD:COUPon:MEASure:SHORt:FILename <String>

For a fixture modeling with the ISD tool and coupon type OPSHort1x (see `CALCulate:FMODel:ISD:COUPon:TYPE`), this command loads the properties of the Short coupon from a Touchstone file.

Parameters:

<String> Path to the Touchstone file, either absolute or relative to the selected working directory (`CALCulate:FMODel:DIRectory`).

Options: R&S ZNB-K220

Manual operation: See "Load File" on page 718

CALCulate:FMODel:ISD<Ph_pt>:COUPon[:STATe] <Boolean>

For a fixture modeling with the ISD tool, this command allows you to specify the ports to which the test coupon is connected.

Must be defined before measuring the test coupon (using `CALCulate:FMODel:ISD:COUPon:MEASure` or `CALCulate:FMODel:ISD:COUPon:MEASure:OPEN` and `CALCulate:FMODel:ISD:COUPon:MEASure:SHORt`).

Suffix:

<Ph_pt> Physical port number

Parameters:

<Boolean> 1 (ON, true) if the test coupon is connected to port <Ph_pt>, 0 (OFF, false) otherwise.

Example: See Section 8.2.8, "Fixture modeling", on page 1685

Options: R&S ZNB-K220

Manual operation: See "Active" on page 718

CALCulate:FMODel:ISD:COUPon:TYPE <CouponType>

Sets/gets the coupon type to be measured for a fixture modeling with the ISD tool.

Parameters:
<CouponType> SYMMetric2x | OPEN1x | SHORt1x | OPSHort1x

Example: See Section 8.2.8, "Fixture modeling", on page 1685

Options: R&S ZNB-K220

Manual operation: See "Coupon Type" on page 717

CALCulate:FMODel:ISD:DCEXtrapolat <Boolean>

Defines whether the deembedding file generated by the ISD tool contains (extrapolated) DC data.

ON corresponds to `add_dc=1` in the tool's batch mode.

Parameters:
<Boolean>

OFF (0)
No DC data is added, i.e. the touchstone data reach down to the start frequency of the instrument sweep.

ON (1)
The ISD tool adds (extrapolated) S-parameter data for f = 0.

*RST: OFF (0)

Options: R&S ZNB-K220

Manual operation: See "DC Extrapolation" on page 722

CALCulate:FMODel:ISD:DIFFonly <Boolean>

Enables the "Differential ony" option of the ISD tool.

Use this option to remove common mode interference, which reduces the higher frequency resonance effect.

Parameters:
<Boolean>

OFF (0)
"Differential ony" disabled (default)

ON (1)
"Differential ony" enabled

Options: R&S ZNB-K220

Manual operation: See "Differential Only" on page 721

CALCulate:FMODel:ISD:DUT:MEASure

For a fixture modeling with the ISD tool, this command allows you to measure DUT + Fixture at the active ports (see `CALCulate:FMODel:ISD<Ph_pt>:DUT[:STATe]`)

Example: See Section 8.2.8, "Fixture modeling", on page 1685

Usage: Event

Options: R&S ZNB-K220

Manual operation: See "Measure" on page 719

CALCulate:FMODel:ISD:DUT:MEASure:FILename <String>

For a fixture modeling with the ISD tool, this command loads the "DUT + Test Fixture" properties from a Touchstone file.

Parameters:
<String> Path to the Touchstone file, either absolute or relative to the selected working directory (`CALCulate:FMODel:DIRectory`).

Options: R&S ZNB-K220

Manual operation: See "Load File" on page 719

CALCulate:FMODel:ISD:DUT:MEASure:FILename:CLEar

Invalidates the file path previously set using `CALCulate:FMODel:ISD:DUT:MEASure:FILename`.

Usage: Event

Options: R&S ZNB-K220

Manual operation: See "Load File" on page 719

CALCulate:FMODel:ISD<Ph_pt>:DUT[:STATe] <Boolean>

For a fixture modeling with the ISD tool, this command allows you to specify the ports to which the test fixture is connected.

Must be defined before measuring DUT + Fixture (using `CALCulate:FMODel:ISD:DUT:MEASure`).

Suffix:
<Ph_pt> Physical port number

Parameters:
<Boolean> 1 (ON, true) if the test fixture is connected to port <Ph_pt>, 0 (OFF, false) otherwise.

Example: See Section 8.2.8, "Fixture modeling", on page 1685

Options: R&S ZNB-K220

Manual operation: See "Active" on page 719

CALCulate:FMODel:ISD:DUT:TYPE <DUTType>

Defines whether the DUT is active or passive.

Corresponds to the `active_dut` batch mode parameter of the ISD tool.

Parameters:
<DUTType> PASSive | ACTive

Options: R&S ZNB-K220

Manual operation: See "DUT Type" on page 723

CALCulate:FMODel:ISD:FTIMe:DUT <FltDutAndLeadIns>

If `CALCulate:FMODel:ISD:FTIMe:OVERride` is set to TRUE, this command allows you to set the flight time for DUT + Lead-ins manually.

Equivalent to setting the `leadin_dut_time` batch mode parameter of the ISD tool to a numeric value (not `auto`).

Parameters:
<FltDutAndLeadIns> Default unit: s

Options: R&S ZNB-K220

Manual operation: See "DUT + lead-ins flight time" on page 723

CALCulate:FMODel:ISD:FTIMe:OVERride <Boolean>

If set to true, the flight time for DUT + lead-in is specified using `CALCulate:FMODel:ISD:FTIMe:DUT`. Otherwise, it is calculated automatically.

True is equivalent to setting the `leadin_dut_time` batch mode parameter of the ISD tool to `auto`.

Parameters:
<Boolean>

Options: R&S ZNB-K220

Manual operation: See "Auto. flight time for DUT + lead-Ins" on page 723

CALCulate:FMODel:ISD:IMPedance <Boolean>

Enables/disables the impedance correction of the ISD tool.

This is a global setting.

Parameters:
<Boolean>

ON (1)
Impedance correction enabled

OFF (0)
Impedance correction disabled

Options: R&S ZNB-K220

Manual operation: See "Impedance Correction" on page 716

CALCulate:FMODel:ISD:OPERation <Operation>

Defines the execution mode of the ISD tool.

Parameters:

<Operation> FAST | ACCurate

ACCurate
Normal execution mode

FAST
Reduces the execution time to ~50%.

Often the FAST mode can be enabled with only little loss of accuracy.

Options: R&S ZNB-K220

Manual operation: See "Operation (fast/acc)" on page 721

CALCulate:FMODel:ISD:PASSivity <Boolean>

Defines whether the ISD tool enforces passivity and reciprocity for the test coupons and the test fixture.

Corresponds to the `passive` batch mode parameter of the ISD tool.

Parameters:

<Boolean> ON (1): enforce passivity and reciprocity
OFF (0): do not enforce passivity and reciprocity

Options: R&S ZNB-K220

Manual operation: See "Enforce Passitivity" on page 721

CALCulate:FMODel:ISD:PORT:ORDer <PortOrder>

Defines how the ISD tool interprets the DUT + Fixture data (see `CALCulate:FMODel:ISD:DUT:MEASure`).

Corresponds to the `port_order` batch mode parameter of the ISD tool.

Parameters:

<PortOrder> ODD | NON | ALL

NON
Ports 1 to N are on the left and ports N+1 to 2*N are on the right.

ODD
Ports 1, 3, 5, etc. are on the left and ports 2, 4, 6, etc. are on the right.

ALL
All ports are on the left (i.e. assumed to be coupled)

Options: R&S ZNB-K220

Manual operation: See "Port Sequence" on page 723

CALCulate:FMODel:ISD:PORT:SKIP <String>

Tells the ISD tool the ports (in the measured "DUT + Test Fixture" file) to be skipped when the tool is run.

Equivalent to using the `ports_to_skip` batch mode parameter of the ISD tool with a list of (positive) port numbers.

Parameters:
<String> Port numbers, separated by blanks (e.g. '1 3 4').

Options: R&S ZNB-K220

Manual operation: See "Ports to Skip (manual)" on page 724

CALCulate:FMODel:ISD:PORT:SKIP:LEFT
CALCulate:FMODel:ISD:PORT:SKIP:NONE
CALCulate:FMODel:ISD:PORT:SKIP:RIGHt

Tells the ISD tool the ports (in the measured DUT + Test Fixture file) to be skipped when the tool is run.

- **...:LEFT**: skip the ports on the left (according to the port order specified using `CALCulate:FMODel:ISD:PORT:ORDer`)
- **...:NONE**: do not skip any ports
- **...:RIGHt**: skip the ports on the right (according to the port order specified using `CALCulate:FMODel:ISD<Ph_pt>:PORT:ORDer`)

Equivalent to setting the `ports_to_skip` batch mode parameter of the ISD tool to -1, 0 or -2, respectively.

Use `CALCulate:FMODel:ISD:PORT:SKIP` to define an arbitrary set of ports to be skipped.

Usage: Event

Options: R&S ZNB-K220

Manual operation: See "Ports to Skip" on page 724

CALCulate:FMODel:ISD:PRESet

Restores the global default settings of the ISD tool.

Usage: Event

Options: R&S ZNB-K220

Manual operation: See "Reset to Default" on page 716

CALCulate:FMODel:ISD:RUN:RUN

Runs the ISD tool.

Before executing this command, make sure that:

- The test coupon measurement (using `CALCulate:FMODel:ISD:COUPon:MEASure:OPEN` and `CALCulate:FMODel:ISD:COUPon:MEASure:SHORt` for 1xOpen+1xShort, `CALCulate:FMODel:ISD:COUPon:MEASure` otherwise) finished successfully, or the test coupon data were successfully loaded from file (using `CALCulate:FMODel:ISD:COUPon:MEASure:FILename`
- The measurement of DUT + test fixture (using `CALCulate:FMODel:ISD:DUT:MEASure`) finished successfully

The resulting Touchstone files are written to C:\Users\Public\Documents\Rohde-Schwarz\Vna\Embedding. If result files with the same name already exist, they are overwritten.

Example: See Section 8.2.8, "Fixture modeling", on page 1685

Usage: Event

Options: R&S ZNB-K220

Manual operation: See "Run <Fixture Modeling Tool>" on page 719

CALCulate:FMODel:ISD<Pt>:RUN[:STATe] <Boolean>

For a fixture modeling with the ISD tool, this command allows you to specify the ports to which the deembedding files are assigned after the tool has run.

Parameters:
<Boolean> 1 (ON) if a deembedding file is assigned to port <Pt>, 0 (OFF) otherwise.

Example: See Section 8.2.8, "Fixture modeling", on page 1685

Options: R&S ZNB-K220

Manual operation: See "Apply" on page 720

CALCulate:FMODel:ISD:SCALe:ATTenuation <AttenLeadInScaling>

Scales the test coupon's attenuation (dB).

Sets/gets the `atten_scale 1` batch mode parameter of the ISD tool

Parameters:
<AttenLeadInScaling>

Options: R&S ZNB-K220

Manual operation: See "Scaling for attenuation" on page 722

CALCulate:FMODel:ISD:SCALe:FREQuency <MaxFrequency>

Defines the maximum frequency to deembed.

Corresponds to the `max_frequency` batch mode parameter of the ISD tool.

Parameters:
<MaxFrequency> Default unit: Hz

Options: R&S ZNB-K220

Manual operation: See "Max Freq to Deembed" on page 722

CALCulate:FMODel:ISD:SCALe:FTIMe <Scaling>

Overrides the lead-in's flight time in case the through-trace test coupon is a bit too short or too long.

Depending on the `CALCulate:FMODel:ISD:SCALe:FTIMe:TYPE`, the value is specified either in relative (default) or in absolute terms.

In relative terms, the command sets/gets the `atten_scale` batch mode parameter of the ISD tool.

Parameters:
<Scaling>
Range: 0.5 to 2.0 for RELative, -100e-9 to 100e-9 for ABSolute scaling type
Increment: 0.01 for RELative, 1e-13 for ABSolute scaling type
*RST: 1 (and RELative scaling type)

Options: R&S ZNB-K220

Manual operation: See "Flight Time Type/Scaling for flight time" on page 723

CALCulate:FMODel:ISD:SCALe:FTIMe:TYPE <ScalingType>

Determines whether `CALCulate:FMODel:ISD:SCALe:FTIMe` specifies the flight time scaling in relative or in absolute terms.

Parameters:
<ScalingType> ABSolute | RELative
*RST: RELative

Options: R&S ZNB-K220

Manual operation: See "Flight Time Type/Scaling for flight time" on page 723

CALCulate:FMODel:ISD:SMALlfixture <Boolean>

Enables/disables the ISD tool's small fixture mode.

Parameters:
<Boolean>

Options: R&S ZNB-K220

Manual operation: See "Small Fixture" on page 722

CALCulate:FMODel:ISD:SMOThing <SMOOTHING>

Defines the smoothing level parameter that is passed to the ISD tool.

For detailed information, see the ISD user guide.

Parameters:

<SMOOTHING> NONE | SMO1 | SMO2 | SMO3 | SMO4

"None", "Smooth," "Smoother", "Smoothest" or "Smoothest + DUT's IL and RL"

Options: R&S ZNB-K220

Manual operation: See "Smoothing" on page 722

CALCulate:FMODel:ISD:TRACe:COUPling <TraceCoupling>

Tells the ISD tool about coupling among lead-in traces.

Corresponds to the `coupling` batch mode parameter of the ISD tool.

Parameters:

<TraceCoupling> NONE | WEAK | STRong

NONE: no coupling.
WEAK: coupling will be extracted even if there are 2 ports enabled on the test coupon
STRong: if the test coupon is a 4-port file, and there are two ports to be extracted, the ISD tool will optimize odd- and even-mode insertion losses

Manual operation: See "Trace Coupling" on page 721

CALCulate:FMODel:ISD:VERSion?

Queries the installed version of the ISD tool.

Usage: Query only

Options: R&S ZNB-K220

Manual operation: See "Version" on page 730

CALCulate:FMODel:SFD...

Commands for PacketMicro's *Smart Fixture De-embedding* (SFD), see https://www.packetmicro.com/Products/sfd-tool.html

CALCulate:FMODel:SFD:AUTO 1008
CALCulate:FMODel:SFD:COUPon:MEASure 1008
CALCulate:FMODel:SFD:COUPon:MEASure:FILename 1008
CALCulate:FMODel:SFD:COUPon:MEASure:FILename:CLEar 1009
CALCulate:FMODel:SFD<Ph_pt>:COUPon[:STATe] 1009
CALCulate:FMODel:SFD:COUPon:TYPE 1009
CALCulate:FMODel:SFD:DIFFcfg 1009

CALCulate:FMODel:SFD:DUT:MEASure 1010
CALCulate:FMODel:SFD:DUT:MEASure:FILename 1010
CALCulate:FMODel:SFD:DUT:MEASure:FILename:CLEar 1010
CALCulate:FMODel:SFD<Ph_pt>:DUT[:STATe] 1010
CALCulate:FMODel:SFD:IMPedance 1011
CALCulate:FMODel:SFD:PRESet 1011
CALCulate:FMODel:SFD:RUN:RUN 1011
CALCulate:FMODel:SFD<Ph_pt>:RUN[:STATe] 1012
CALCulate:FMODel:SFD:TOTaldiffcfg 1012
CALCulate:FMODel:SFD:VERSion? 1012

CALCulate:FMODel:SFD:AUTO <Boolean>

Enables/disables the impedance correction of the SFD tool.

This is a global setting.

Duplicate of `CALCulate:FMODel:SFD:IMPedance`, kept for backward compatibility.

Parameters:

<Boolean>
ON (1)
Impedance correction enabled
OFF (0)
Impedance correction disabled

Options: R&S ZNB-K230

Manual operation: See "Impedance Correction" on page 716

CALCulate:FMODel:SFD:COUPon:MEASure

For a fixture modeling with the SFD tool, this command starts the measurement of the coupon at the active ports (see `CALCulate:FMODel:SFD<Ph_pt>:COUPon[:STATe]`).

The coupon type can be selected using `CALCulate:FMODel:SFD:COUPon:TYPE`.

Usage: Event

Options: R&S ZNB-K230

Manual operation: See "Measure" on page 717

CALCulate:FMODel:SFD:COUPon:MEASure:FILename <String>

For a fixture modeling with the SFD tool, this command loads the coupon properties from a Touchstone file.

The coupon type can be selected using `CALCulate:FMODel:SFD:COUPon:TYPE`.

Parameters:

<String>
Path to the Touchstone file, either absolute or relative to the selected working directory (`CALCulate:FMODel:DIRectory`).

Options: R&S ZNB-K230

Manual operation: See "Load File" on page 718

CALCulate:FMODel:SFD:COUPon:MEASure:FILename:CLEar

Invalidates the file path previously set using `CALCulate:FMODel:SFD:COUPon:MEASure:FILename`.

Usage: Event

Options: R&S ZNB-K230

Manual operation: See "Load File" on page 718

CALCulate:FMODel:SFD<Ph_pt>:COUPon[:STATe] <Boolean>

For a fixture modeling with the SFD tool, this command allows you to specify the ports to which the test coupon is connected.

Must be defined before measuring the test coupon (using `CALCulate:FMODel:SFD:DUT:MEASure`).

Suffix:
<Ph_pt> Physical port number

Parameters:
<Boolean> 1 (ON, true) if the test coupon is connected to port <Ph_pt>, 0 (OFF, false) otherwise.

Options: R&S ZNB-K230

Manual operation: See "Active" on page 718

CALCulate:FMODel:SFD:COUPon:TYPE <CouponType>

Sets/gets the coupon type to be measured for a fixture modeling with the SFD tool.

Parameters:
<CouponType> SYMMetric2x | OPEN1x | SHORt1x

Options: R&S ZNB-K230

Manual operation: See "Coupon Type" on page 717

CALCulate:FMODel:SFD:DIFFcfg <SFDPortConfig>

Tells the SFD tool about the port ordering of the 2x Thru test coupon.

Parameters:
<SFDPortConfig> ODD |

ODD
Odd ports are on the left and even ports are on the right.

NON
Ports 1 to N are on the left and ports N+1 to 2·N are on the right.

Options: R&S ZNB-K230

Manual operation: See "2x Thru Port Ordering" on page 724

CALCulate:FMODel:SFD:DUT:MEASure

For a fixture modeling with the SFD tool, this command allows you to measure the DUT + Fixture at the active ports (see `CALCulate:FMODel:SFD<Ph_pt>:DUT[:STATe]`).

Usage: Event

Options: R&S ZNB-K230

Manual operation: See "Measure" on page 719

CALCulate:FMODel:SFD:DUT:MEASure:FILename <String>

For a fixture modeling with the SFD tool, this command loads the "DUT + Test Fixture" properties from a Touchstone file.

Parameters:
<String> Path to the Touchstone file, either absolute or relative to the selected working directory (`CALCulate:FMODel:DIRectory`).

Options: R&S ZNB-K230

Manual operation: See "Load File" on page 719

CALCulate:FMODel:SFD:DUT:MEASure:FILename:CLEar

Invalidates the file path previously set using `CALCulate:FMODel:SFD:DUT:MEASure:FILename`.

Usage: Event

Options: R&S ZNB-K230

Manual operation: See "Load File" on page 719

CALCulate:FMODel:SFD<Ph_pt>:DUT[:STATe] <Boolean>

For a fixture modeling with the SFD tool, this command allows you to specify the ports to which the test fixture is connected.

Must be defined before measuring DUT + Fixture (using `CALCulate:FMODel:SFD:DUT:MEASure`).

Suffix:
<Ph_pt> Physical port number

Parameters:

<Boolean> 1 (ON, true) if the test fixture is connected to port <Ph_pt>, 0 (OFF, false) otherwise.

Options: R&S ZNB-K230

Manual operation: See "Active" on page 719

CALCulate:FMODel:SFD:IMPedance <Boolean>

Enables/disables the impedance correction of the SFD tool.

This is a global setting.

Parameters:

<Boolean>

ON (1)
Impedance correction enabled

OFF (0)
Impedance correction disabled

Options: R&S ZNB-K230

Manual operation: See "Impedance Correction" on page 716

CALCulate:FMODel:SFD:PRESet

Restores the global default settings of the SFD tool.

Usage: Event

Options: R&S ZNB-K230

Manual operation: See "Reset to Default" on page 716

CALCulate:FMODel:SFD:RUN:RUN

Runs the SFD tool.

Before executing this command, make sure that:

- The test coupon measurement (using `CALCulate:FMODel:SFD:COUPon:MEASure`) finished successfully, or the test coupon data were successfully loaded from file (using `CALCulate:FMODel:SFD:COUPon:MEASure:FILename`
- The measurement of DUT + test fixture (using `CALCulate:FMODel:SFD:DUT:MEASure`) finished successfully.

The resulting Touchstone files are written to C:\Users\Public\Documents\Rohde-Schwarz\Vna\Embedding. If result files with the same name already exist, they are overwritten.

Usage: Event

Options: R&S ZNB-K230

Manual operation: See "Run <Fixture Modeling Tool>" on page 719

CALCulate:FMODel:SFD<Ph_pt>:RUN[:STATe] <Boolean>

For a fixture modeling with the SFD tool, this command allows you to specify the ports to which the deembedding files are assigned after the tool has run.

Suffix:
<Ph_pt> Physical port number for single-ended deembedding, logical port number for balanced deembedding.

Parameters:
<Boolean> 1 (ON) if a deembedding file is assigned to port <Pt>, 0 (OFF) otherwise.

Options: R&S ZNB-K230

Manual operation: See "Apply" on page 720

CALCulate:FMODel:SFD:TOTaldiffcfg <SFDPortConfig>

Tells the SFD tool about the port ordering of the test fixture.

Parameters:
<SFDPortConfig> ODD | NON

ODD
Odd ports are on the left and even ports are on the right.

NON
Ports 1 to N are on the left and ports N+1 to 2·N are on the right.

Options: R&S ZNB-K230

Manual operation: See "Total Port Ordering" on page 725

CALCulate:FMODel:SFD:VERSion?

Queries the installed version of the SFD tool.

Usage: Query only

Manual operation: See "Version" on page 730

CALCulate:FMODel... (other)

CALCulate:FMODel:COPY:CHANnel:DEEMbedding:BALanced....1013
CALCulate:FMODel:COPY:CHANnel:DEEMbedding:PSET....1013
CALCulate:FMODel:COPY:CHANnel:DEEMbedding:SINGleended....1013
CALCulate:FMODel:DIRectory....1013
CALCulate:FMODel:DIRectory:DEFault....1013
CALCulate:FMODel:DIRectory:DEFault:CLEar....1013
CALCulate:FMODel:QSETup....1013
CALCulate:FMODel:REName....1014

CALCulate:FMODel:COPY:CHANnel:DEEMbedding:BALanced <Channel Id From>, <Channel Id To>
CALCulate:FMODel:COPY:CHANnel:DEEMbedding:PSET <Channel Id From>, <Channel Id To>
CALCulate:FMODel:COPY:CHANnel:DEEMbedding:SINGleended <From>, <To>

Copies the respective deembedding settings ("Balanced", "Port Sets", "Single Ended")from channel no. `<From>` to channel no. `<To>`.

Setting parameters:

<From>, <To> Channel numbers

Usage: Setting only

Manual operation: See "Copy Deembedding to Other Channels" on page 704

CALCulate:FMODel:DIRectory <String>

Defines the common working directory of the ISD, SFD and EZD fixture modeling tools.

Parameters:

<String> Directory path, either absolute or relative to the current directory (`MMEMory:CDIRectory`).

Manual operation: See "Set Directory/Timestamp Filenames" on page 715

CALCulate:FMODel:DIRectory:DEFault

Resets the common working directory of the ISD, SFD, and EZD tool to its default `C:\Users\Public\Documents\Rohde-Schwarz\Vna\Embedding`.

Usage: Event

Manual operation: See "Set Directory/Timestamp Filenames" on page 715

CALCulate:FMODel:DIRectory:DEFault:CLEar

Deletes all files in the common default working directory of the of the ISD, SFD, and EZD tool (`C:\Users\Public\Documents\Rohde-Schwarz\Vna\Embedding`).

Usage: Event

Manual operation: See "Set Directory/Timestamp Filenames" on page 715

CALCulate:FMODel:QSETup

The query returns the recommended sweep parameters for fixture modeling with the selected tool (see `CALCulate:FMODel:DEASsistant:TOOL`), the setting applies the recommended values.

The query returns a comma-separated list of numeric values: Start Frequency (Hz), Stop Frequency (Hz), Span (Hz), Sweep Points

Manual operation: See "Recommended/Use Recommended Values" on page 763

CALCulate:FMODel:REName <Boolean>

If set to ON (1), the names of subsequently generated "Test Coupon" and "DUT + Test Fixture" files are prefixed with the current date and time.

Parameters:
<Boolean>

Manual operation: See "Set Directory/Timestamp Filenames" on page 715

7.3.1.7 CALCulate:LIMit...

The `CALCulate:LIMit...` commands define the limit lines and control the limit check.

CALCulate<Chn>:LIMit:CIRCle:CLEar....1015
CALCulate<Chn>:LIMit:CIRCle:DATA....1015
CALCulate<Chn>:LIMit:CIRCle:DISPlay[:STATe]....1015
CALCulate<Chn>:LIMit:CIRCle:FAIL?....1016
CALCulate:LIMit:CIRCle:FAIL:ALL?....1016
CALCulate<Chn>:LIMit:CIRCle:SOUNd[:STATe]....1016
CALCulate<Chn>:LIMit:CIRCle[:STATe]....1017
CALCulate<Chn>:LIMit:CLEar....1017
CALCulate<Chn>:LIMit:CONTrol[:DATA]....1017
CALCulate<Chn>:LIMit:CONTrol:SHIFt....1018
CALCulate<Chn>:LIMit:DATA....1019
CALCulate<Chn>:LIMit:DCIRcle:CLEar....1020
CALCulate<Chn>:LIMit:DCIRcle:DATA....1020
CALCulate<Chn>:LIMit:DCIRcle:DISPlay[:STATe]....1020
CALCulate<Chn>:LIMit:DCIRcle[:STATe]....1020
CALCulate<Chn>:LIMit:DELete:ALL....1021
CALCulate<Chn>:LIMit:DISPlay[:STATe]....1021
CALCulate<Chn>:LIMit:FAIL?....1021
CALCulate:LIMit:FAIL:ALL?....1022
CALCulate:LIMit:FAIL:DATA?....1022
CALCulate<Chn>:LIMit:LOWer[:DATA]....1022
CALCulate<Chn>:LIMit:UPPer[:DATA]....1022
CALCulate<Chn>:LIMit:LOWer:FEED....1024
CALCulate<Chn>:LIMit:UPPer:FEED....1024
CALCulate<Chn>:LIMit:LOWer:SHIFt....1025
CALCulate<Chn>:LIMit:UPPer:SHIFt....1025
CALCulate:LIMit:POINts:LOWer?....1025
CALCulate:LIMit:POINts:UPPer?....1025
CALCulate<Chn>:LIMit:SEGMent<Seg>:AMPLitude:STARt....1026
CALCulate<Chn>:LIMit:SEGMent<Seg>:AMPLitude:STOP....1026
CALCulate<Chn>:LIMit:SEGMent:COUNt?....1027
CALCulate<Chn>:LIMit:SEGMent<Seg>:FORMula....1027
CALCulate<Chn>:LIMit:SEGMent<Seg>:FORMula:STATe....1028

CALCulate<Chn>:LIMit:SEGMent<Seg>:INTerpol 1028
CALCulate<Chn>:LIMit:SEGMent<Seg>:STIMulus:STARt 1029
CALCulate<Chn>:LIMit:SEGMent<Seg>:STIMulus:STOP 1029
CALCulate<Chn>:LIMit:SEGMent<Seg>:TYPE 1029
CALCulate<Chn>:LIMit:SOUNd[:STATe] 1030
CALCulate<Chn>:LIMit:STATe 1030
CALCulate<Chn>:LIMit:STATe:AREA 1031
CALCulate<Chn>:LIMit:TTLout<Pt>[:STATe] 1031
CALCulate<Chn>:LIMit:X:OFFSet 1032
CALCulate<Chn>:LIMit:Y:OFFSet 1032

CALCulate<Chn>:LIMit:CIRCle:CLEar

Resets the circle test for the active trace of channel <Chn>.

Suffix:
<Chn> Channel number used to identify the active trace

Usage: Event

Manual operation: See "Clear Test" on page 500

CALCulate<Chn>:LIMit:CIRCle:DATA <CenterX>, <CenterY>, <Radius>

Defines a circle limit line by its center coordinates and its radius.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<CenterX>
Range: Virtually no restriction for center coordinates.
*RST: 0
Default unit: NN

<CenterY>
Range: Virtually no restriction for center coordinates.
*RST: 0
Default unit: NN

<Radius>
Range: Virtually no restriction for radius (use positive values).
*RST: 1
Default unit: NN

Example: See `CALCulate<Chn>:LIMit:CIRCle[:STATe]`

Manual operation: See "Radius / Center X / Center Y" on page 500

CALCulate<Chn>:LIMit:CIRCle:DISPlay[:STATe] <Boolean>

Displays or hides the circle limit line associated to the active trace.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:

<Boolean> ON | OFF - Circle limit line on or off.

*RST: OFF

Example: `*RST; CALCulate:LIMit:CIRCle:DATA 0, 0, 0.5`
Define a circle limit line, centered at the origin of the polar diagram, assigning a radius of 0.5 U.
`CALCulate:FORMat POLar`
`CALCulate:LIMit:CIRCle:DISPlay ON`
Activate a polar diagram and show the circle limit line in the diagram.

Manual operation: See "Show Limit Circle" on page 498

CALCulate<Chn>:LIMit:CIRCle:FAIL?

Returns a 0 or 1 to indicate whether or not the circle limit check has failed. 0 represents pass and 1 represents fail

Tip: Use `CALCulate:CLIMits:FAIL?` to perform a composite (global) limit check.

Suffix:

<Chn> Channel number used to identify the active trace

Example: See `CALCulate<Chn>:LIMit:CIRCle[:STATe]`

Usage: Query only

Manual operation: See "Limit Check" on page 498

CALCulate:LIMit:CIRCle:FAIL:ALL? [<RecallSet>]

Returns a 0 or 1 to indicate whether or not the circle limit check has failed for at least one channel in the referenced recall set. 0 represents pass and 1 represents fail

Query parameters:

<RecallSet> Recall set name; if omitted the active recall set is used

Usage: Query only

Manual operation: See "Limit Check" on page 498

CALCulate<Chn>:LIMit:CIRCle:SOUNd[:STATe] <Boolean>

Deprecated command: Since version 3.45, generating sounds is no longer supported by the analyzer firmware.

Switches the acoustic signal (fail beep) on or off. The fail beep is generated each time the analyzer detects an exceeded circle limit.

Suffix:

<Chn> Channel number used to identify the active trace

Parameters:
<Boolean> ON | OFF - Fail beep on or off.
*RST: OFF

Example: `CALCulate:LIMit:CIRCle:STATe ON; SOUNd ON`
Switch on the limit check and activate the fail beep.

CALCulate<Chn>:LIMit:CIRCle[:STATe] <Boolean>

Switches the circle limit check on or off.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<Boolean>

Example: `*RST; CALCulate:LIMit:CIRCle:DATA 0, 0, 0.5`
Define a circle limit line, centered at the origin of the polar diagram, assigning a radius of 0.5 U.
`CALCulate:LIMit:CIRCle:STATe ON; FAIL?`
Switch on the limit check and query the result.

Manual operation: See "Limit Check" on page 498

CALCulate<Chn>:LIMit:CLEar

Resets the limit check results for the limit line test.

Suffix:
<Chn> Channel number used to identify the active trace

Usage: Event

Manual operation: See "Clear Test" on page 486

CALCulate<Chn>:LIMit:CONTrol[:DATA] <StartStim>, <StopStim>[, ...]

Defines the stimulus values of the limit line and/or creates limit line segments. See also Section 4.4.1.1, "Rules for limit line definition", on page 136.

Rules for creating segments

The following rules apply to an active trace with n existing limit line segments:

- An odd number of values is rejected; an error message -109,"Missing parameter..." is generated.
- An even number of 2*k values updates or generates k limit line segments.
- For n > k, the stimulus values of all existing limit line segments no. 1 to k are updated, the existing limit line segments no. k+1, ..., n are deleted.
- For n < k, the stimulus values of the limit line segments no. 1 to n are updated. The limit line segments n+1, ..., k are generated with default response values (see

CALCulate<Chn>:LIMit:UPPer[:DATA], CALCulate<Chn>:LIMit:LOWer[:DATA]).

Note: The generated segments are upper or lower limit line segments, depending on the CALCulate<Chn>:LIMit:SEGMent<Seg>:TYPE setting.
CALCulate<Ch>:LIMit:CONTrol[:DATA] does not overwrite the type setting.

Tip: To define additional new limit line segments without overwriting the old segments, use CALCulate<Chn>:LIMit:DATA.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<StartStim>

<StopStim> Pairs of stimulus values, each pair confining a limit line segment.
See also Section 4.4.1.1, "Rules for limit line definition", on page 136.
If not specified, the units are adjusted to the sweep type of the active channel ([SENSe<Ch>:]SWEep:TYPE).

Range: Virtually no restriction for limit segments.
*RST: A segment that is created implicitly, e.g. by means of CALCulate<Ch>:LIMit:UPPer[:DATA] or CALCulate<Ch>:LIMit:LOWer[:DATA,], covers the maximum sweep range of the analyzer.

Example: *RST; :CALC:LIM:CONT 1 GHZ, 2 GHZ
Select a linear frequency sweep (default) and define an upper limit line segment in the stimulus range between 1 GHz and 2 GHz, using default response values (–40 dB).
CALC:LIM:DISP ON
Show the limit line segment in the active diagram.

Manual operation: See "Segment List" on page 489

CALCulate<Chn>:LIMit:CONTrol:SHIFt <LimShift>

Shifts all limit line segments of channel <Chn>'s active trace in horizontal direction.

Use CALCulate<Chn>:LIMit:UPPer:SHIFt and CALCulate<Chn>:LIMit:LOWer:SHIFt to shift the line segments in vertical direction.

Suffix:
<Chn> Channel number used to identify the active trace

Setting parameters:
<LimShift> Offset value for the limit line

Range: Virtually no restriction for limit segments
Default unit: NN

Example: `*RST; :CALC:LIM:CONT 1 GHZ, 2 GHZ`
Define a limit line segment in the stimulus range between 1 GHz and 2 GHz, using default response values.
`CALC:LIM:CONT:SHIF 1; :CALC:LIM:CONT?`
Shift the segment by 1 Hz. The modified limit line segment ranges from `1000000001` (Hz) to `2000000001` (Hz).

Usage: Setting only

Manual operation: See "Segment List" on page 489

CALCulate<Chn>:LIMit:DATA <Type>, <StartStim>, <StopStim>, <StartResp>, <StopResp>

Defines the limit line type, the stimulus and response values for a limit line with an arbitrary number of limit line segments. See Section 4.4.1.1, "Rules for limit line definition", on page 136.

Note: In contrast to `CALCulate<Chn>:LIMit:CONTrol[:DATA]`, this command does not overwrite existing limit line segments. The defined segments are appended to the segment list as new segments.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<Type> Identifier for the type of the limit line segment:
0 – limit line segment off, segment defined but no limit check performed.
1 – upper limit line segment
2 – lower limit line segment
Range: 0, 1, 2 (see above)

<StartStim>, <StopStim>, <StartResp>, <StopResp> Stimulus and response values of the first and last points of the limit line segment.
The unit of the stimulus values is adjusted to the sweep type of the active channel (`[SENSe<Ch>:]SWEep:TYPE`). The unit of the ripple limit is adjusted to the format of the active trace (`CALCulate<Chn>:FORMat`).

Example: `*RST; :CALC:LIM:CONT 1 GHZ, 1.5 GHZ`
Define an upper limit line segment in the stimulus range between 1 GHz and 1.5 GHz, using default response values.
`CALC:LIM:DATA 1, 1500000000, 2000000000, 2, 3`
Define an upper limit line segment in the stimulus range between 1.5 GHz and 2 GHz, assigning response values of +2 dB and +3 dB.
`CALC:LIM:DISP ON`
Show the limit line segment in the active diagram.

Manual operation: See "Segment List" on page 489

CALCulate<Chn>:LIMit:DCIRcle:CLEar

Resets the display circle to its default configuration (unit circle; show border: off; limit to circle: off).

Suffix:
<Chn> Channel number used to identify the active trace

Usage: Event

Manual operation: See "Clear Circle" on page 502

CALCulate<Chn>:LIMit:DCIRcle:DATA <CenterX>, <CenterY>, <Radius>

Defines the display circle for the active trace of channel <Chn>.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<CenterX> X position (real part) of the display circle's center

<CenterY> Y position (imaginary part) of the display circle's center

<Radius> Radius of the display circle

Manual operation: See "Draw Circle / Radius, Center X, Center Y" on page 502

CALCulate<Chn>:LIMit:DCIRcle:DISPlay[:STATe] <Boolean>

Sets/queries the visibility of the display circle for the active trace of channel <Chn>.

The display circle is defined using `CALCulate<Chn>:LIMit:DCIRcle:DATA`.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<Boolean> If set to ON, the line of the display circle is shown whenever the related trace is displayed in complex format.

Manual operation: See "Show Border" on page 501

CALCulate<Chn>:LIMit:DCIRcle[:STATe] <Boolean>

Sets/queries the state of the display circle for the active trace of channel <Chn>.

The display circle is defined using `CALCulate<Chn>:LIMit:DCIRcle:DATA`.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<Boolean> If set to ON, only trace points within the display circle are shown at the GUI whenever the related trace is displayed in complex format.

Manual operation: See "Limit to Circle On/Off" on page 501

CALCulate<Chn>:LIMit:DELete:ALL

Deletes all limit line segments.

Suffix:
<Chn> Channel number used to identify the active trace

Example:
```
*RST; :CALC:LIM:CONT 1 GHZ, 1.5 GHZ
```
Define an upper limit line segment in the stimulus range between 1 GHz and 1.5 GHz, using default response values.
```
CALC:LIM:DATA 1,1500000000, 2000000000,2,3
```
Define an upper limit line segment in the stimulus range between 1.5 GHz and 2 GHz, assigning response values of +2 dB and +3 dB.
```
CALC:LIM:DEL:ALL
```
Delete both created limit line segments.

Usage: Event

Manual operation: See "Add / Insert / Delete / Delete All" on page 490

CALCulate<Chn>:LIMit:DISPlay[:STATe] <Boolean>

Displays or hides the entire limit line (including all segments) associated to the active trace.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<Boolean> ON | OFF - Limit line on or off.
*RST: OFF

Example:
```
*RST; :CALC:LIM:CONT 1 GHZ, 2 GHZ
```
Define an upper limit line segment in the stimulus range between 1 GHz and 2 GHz, using default response values.
```
CALC:LIM:DISP ON
```
Show the limit line segment in the active diagram.

Manual operation: See "Show Limit Line" on page 485

CALCulate<Chn>:LIMit:FAIL?

Returns a 0 or 1 to indicate whether or not the limit check has failed. 0 represents pass and 1 represents fail

Tip: Use `CALCulate:CLIMits:FAIL?` to perform a composite (global) limit check.

Since V2.20 of the R&S ZNB FW the result is automatically recalculated whenever a relevant setting is changed, i.e. a subsequent query returns the updated limit violation state.

Suffix:
<Chn> Channel number used to identify the active trace

Example: `*RST; :CALC:LIM:CONT 1 GHZ, 2 GHZ`
Define an upper limit line segment in the stimulus range between 1 GHz and 2 GHz, using default response values.
`CALC:LIM:STAT ON; FAIL?`
Switch on the limit check, and query the result.
`CALC:LIM:STAT:AREA LEFT, TOP`
For a subsequent check at the GUI or a hardcopy, move the pass/fail message to the top-left position.

Usage: Query only

Manual operation: See "Limit Check" on page 485

CALCulate:LIMit:FAIL:ALL? [<RecallSet>]

Returns a 0 or 1 to indicate whether or not the limit line check has failed for at least one channel in the referenced recall set. 0 represents pass and 1 represents fail.

Query parameters:
<RecallSet> Recall set name; if omitted the active recall set is used

Usage: Query only

Manual operation: See "Limit Check" on page 485

CALCulate:LIMit:FAIL:DATA? <TraceName>, <LimitFailType>

Returns those sweep points that have caused a limit violation of the given <LimitFail-Type> for trace <TraceName>.

Query parameters:
<TraceName> Trace name, uniquely identifying the related trace

<LimitFailType> LIMit | RIPPle | CIRCle
Limit (fail) type

Usage: Query only

CALCulate<Chn>:LIMit:LOWer[:DATA] <ResponseValue>, <ResponseValue>...
CALCulate<Chn>:LIMit:UPPer[:DATA] <StartResponse>,
<StopResponse>[,<StartResponse>, <StopResponse>[,...]]

Sets/gets the response (y-axis) values of the lower/upper limit lines and/or creates limit line segments. See also Section 4.4.1.1, "Rules for limit line definition", on page 136.

Note that in contrast to commands addressing a single limit line segment `<Seg>` (such as `CALCulate<Chn>:LIMit:SEGMent<Seg>:TYPE`), these commands **assume** that:

- Lower limit line segments have even numbers (`<Seg> = 2,4,6,...`) and
- Upper limit line segments have odd numbers (`<Seg> = 1,3,5,...`).

`CALCulate<Chn>:LIMit:LOWer/UPPer` sets the type and response values of even/odd limit line segments and gets the response values of even/odd limit line segments - no matter what the current type of these segments actually is

Both commands only work, if the total number of limit line segments is even.

Rules for creating and updating segments

Suppose that the active trace is equipped with 2s limit line segments (of any type) and k pairs of response values are passed with the command.

- `CALCulate:LIMit:LOWer`
  - Deletes "obsolete" limit line segments 2k+1, ..., 2s
  - Updates even limit line segments 2, 4, ..., 2s with type=lower and the given response values
  - Creates lower limit line segments 2s+2, 2s+4, ..., 2k with (type=lower and) the given response values
  - Creates "missing" upper limit line segments 2s+1, 2s+3, ..., 2k-1 with (type=upper and) default response values
- `CALCulate:LIMit:UPPer`
  - Deletes "obsolete" limit line segments 2k+1, ..., 2s
  - Updates odd limit line segments 1, 3, ..., 2s-1 with type=upper and the given response values
  - Creates upper limit line segments 2s+1, 2s+3, ..., 2k-1 with (type=upper and) the given response values
  - Creates "missing" lower limit line segments 2s+2, 2s+4, ..., 2k with (type=lower and) default response values

If s>0, newly created lower/upper limit line segments inherit their start and stop stimuli from the limit line segment with the highest even/odd number. Otherwise their stimulus range is set to the entire sweep range.

See `CALCulate<Chn>:LIMit:CONTrol[:DATA]` on how to change the stimulus values of a limit line segment.

Suffix:

<Chn> Channel number used to identify the active trace

Parameters:

<StartResponse>, <StopResponse>, ... Pai(s of response values. In the parameter list, item 2s-1 defines the "Start Response" and item 2s defines the "Stop Response" of upper/lower limit line segment s = 1,2,...
An odd number of values is rejected with an error message -109,"Missing parameter...".

Range: Virtually no restriction for limit segments.
*RST: Implicitly created segments are created with a default response value of -40dB.
Default unit: dB

Example: `CALC:LIM:LOW -10, 0, 0, -10`
Define two limit line segments covering the entire sweep range. Two upper limit line segments with default response values are created in addition.
`CALC:LIM:UPP 0, 10, 10, 0`
Change the response values of the upper limit line segments .
`CALC:LIM:DISP ON`
Show the limit line segments in the active diagram.

Manual operation: See "Segment List" on page 489

CALCulate<Chn>:LIMit:LOWer:FEED <StimulusOffset>, <ResponseOffset>[, <TraceName>]
CALCulate<Chn>:LIMit:UPPer:FEED <StimulusOffset>, <ResponseOffset>[, <TraceName>]

Generates a lower or an upper limit line using the stimulus values of a data or memory trace and specified offset values.

Suffix:
<Chn> Channel number used to identify the active trace. This trace provides the stimulus data for the limit line unless another trace `<TraceName>` is specified.

Setting parameters:
<StimulusOffset> Stimulus offset value, shifting all imported limit line segments in horizontal direction.
Default unit: NN

<ResponseOffset> Response offset value, shifting all imported limit line segments in vertical direction.
Default unit: dB

<TraceName> Name of the selected trace as used, e.g., in `CALCulate<Ch>:PARameter:SDEFine`. If no trace name is specified the analyzer uses the active trace no. `<Chn>`.

Example: `CALC:LIM:LOW:FEED 1 GHZ, -10`
Use the stimulus values of the active trace, shifted by 1 GHz to the right and decreased by –10 dB, to create a lower limit line.
`CALC:LIM:UPP:FEED 1 GHZ, 10`
Use the stimulus values of the active trace, shifted by 1 GHz to the right and increased by 10 dB, to create an upper limit line.
`CALC:LIM:LOW:SHIF -3; :CALC:LIM:CONT:SHIF 1 GHz`
Shift the lower limit line by an additional -3 dB in vertical and by 1 GHz in horizontal direction. The upper limit line is also shifted.

Usage: Setting only

Manual operation: See "Segment List" on page 489

CALCulate<Chn>:LIMit:LOWer:SHIFt <LimShift>
CALCulate<Chn>:LIMit:UPPer:SHIFt <LimShift>

These commands shift all lower/upper limit line segments of channel <Chn>'s active trace in vertical direction.

Use `CALCulate<Chn>:LIMit:CONTrol:SHIFt` to shift the line segments in horizontal direction.

Suffix:
<Chn> Channel number used to identify the active trace

Setting parameters:
<LimShift> Response offset value for all limit line segments.

Range: Virtually no restriction for limit segments
Default unit: NN

Example: See `CALCulate<Chn>:LIMit:LOWer:FEED`

Usage: Setting only

CALCulate:LIMit:POINts:LOWer? <TraceName>
CALCulate:LIMit:POINts:UPPer? <TraceName>

Reads the effective lower/upper limit points from a limit line test for an arbitrary (not necessarily the active) trace, referenced by its name <TraceName>.

"Effective“ means:

- Only points within the sweep range and only active limit checks are considered.
- Disabled limit line segments (`CALCulate<Chn>:LIMit:SEGMent<Seg>:TYPE OFF`) are ignored.
- Both linear/logarithmic interpolation and limit line formulae are supported.
- If there are overlapping limit line segments, only the tighter limit is reported.

Query parameters:
<TraceName> String parameter containing the trace name

Example:

```
*RST
:SENSe1:FREQuency1:STARt 1 GHZ; STOP 5 GHZ
:SENSe1:SWEep:POINts 5
```

After a reset, modify the default trace 'Trc1'.

```
:CALCulate1:LIMit1:DATA 1, 1000000000,
2000000000, -40, -30
:CALCulate1:LIMit2:DATA 1, 4000000000,
5000000000, -30, -40
```

Define upper limit lines for frequency ranges [1 GHz, 2 GHz] and [4 GHz, 5 GHz]

```
:CALCulate1:LIMit1:POINts:UPPer? 'Trc1'
```

Query the limit lines. Returns an empty string, because limit checking is inactive.

```
:CALCulate1:LIMit1:STATe ON
```

Activate limit checking.

```
:CALCulate17:LIMit1:POINts:UPPer? 'Trc1'
```

Returns `-40,-30,-NAN(IND),-30,-40`, where `-NAN(IND)` is related to the center frequency `3 GHz`, whose trace value is not limited.

Usage: Query only

Manual operation: See "Limit Check" on page 485

CALCulate<Chn>:LIMit:SEGMent<Seg>:AMPLitude:STARt <Response>
CALCulate<Chn>:LIMit:SEGMent<Seg>:AMPLitude:STOP <Response>

These commands change the start or the stop response values (i.e. the response values assigned to the start or stop stimulus values) of a limit line segment. A segment must be created first to enable the commands (e.g `CALCulate<Chn>:LIMit:DATA`). See also Section 4.4.1.1, "Rules for limit line definition", on page 136.

Tip: To define the response values of several limit line segments with a single command, use `CALCulate<Chn>:LIMit:LOWer[:DATA]` or `CALCulate<Chn>:LIMit:UPPer[:DATA]`.

Suffix:

<Chn> Channel number used to identify the active trace

<Seg> Segment number

Parameters:

<Response> Response value

Range: Virtually no restriction for limit segments
*RST: The default response values of a segment that is created by defining its stimulus values only (e.g. by means of CALCulate<Ch>:LIMit:CONTrol[:DATA]), are -40 dB.
Default unit: NN

Example: `CALC:LIM:DATA 1,1500000000, 2000000000,2,3`
Define an upper limit line segment (segment no. 1) in the stimulus range between 1.5 GHz and 2 GHz, assigning response values of +2 dB and +3 dB.
`:CALC:LIM:SEGM:AMPL:STAR 5; STOP 5; :CALC:LIM:SEGM:TYPE LMIN`
Change the segment to a lower limit line segment with a constant response value of +5 dB.
`CALC:LIM:DATA?`
Query the type, the stimulus and response values of the created segment with a single command. The response is 2,1500000000,2000000000,5,5.

Manual operation: See "Segment List" on page 489

CALCulate<Chn>:LIMit:SEGMent:COUNt?

Returns the number of limit line segments, including enabled and disabled segments.

Suffix:
<Chn> Channel number used to identify the active trace

Example: `CALC:LIM:DATA 1,1500000000, 2000000000,2,3`
Define an upper limit line segment (segment no. 1) in the stimulus range between 1.5 GHz and 2 GHz, assigning response values of +2 dB and +3 dB.
`CALC:LIM:SEGM:COUNT?`
Query the number of segments. The response is 1.

Usage: Query only

Manual operation: See "Segment List" on page 489

CALCulate<Chn>:LIMit:SEGMent<Seg>:FORMula <Formula>

Defines a custom formula for limit line segment <Seg> of the related trace.

The formula is only used if `CALCulate<Chn>:LIMit:SEGMent<Seg>:FORMula` is set to `ON`.

Suffix:
<Chn> Channel number used to identify the active trace

<Seg> Segment number

Parameters:
<Formula> User-defined formula (string).
Allows for the same mathematical expressions as `CALCulate<Chn>:MATH[:EXPRession]:SDEFine`.

Example:

```
*RST
SENS:FREQ:STAR 1GHz
SENS:FREQ:STOP 3GHz
CALC:LIM:CONT 1GHz, 2GHz, 2GHz, 3GHz
CALC:LIM:SEGM1:FORM '22-(20/25.78)*StimVal/1e9'
CALC:LIM:SEGM1:FORM:STAT ON
CALC:LIM:SEGM2:FORM '15-(6/25.78)*StimVal/1e9'
CALC:LIM:SEGM2:TYPE LMIN
CALC:LIM:DISP ON
CALC:LIM:STAT ON
```

Manual operation: See "Linear/Formula" on page 492

CALCulate<Chn>:LIMit:SEGMent<Seg>:FORMula:STATe <State>

Defines whether a custom formula is used for limit line segment <Seg> of the related trace.

Suffix:

<Chn> Channel number used to identify the active trace

<Seg> Segment number
If you want to use a custom formula for a segment, use `CALCulate<Chn>:LIMit:CONTrol[:DATA]` to create it.

Parameters:

<State>

OFF (0)
The line segment is a straight line, connecting the endpoints specified in the segment definition.

ON (1)
The line segment is defined by a custom formula. Use `CALCulate<Chn>:LIMit:SEGMent<Seg>:FORMula` to specify the formula.

*RST: OFF

Example: See `CALCulate<Chn>:LIMit:SEGMent<Seg>:FORMula` on page 1027

Manual operation: See "Linear/Formula" on page 492

CALCulate<Chn>:LIMit:SEGMent<Seg>:INTerpol <LimLineInterpolation>

"Interpolation" determines whether the limit line segment is interpolated linearly or logarithmically. The latter can be more suitable for logarithmic sweeps.

Suffix:

<Chn> Channel number used to identify the active trace

<Seg> Segment number

Parameters:

<LimLineInterpolation> LOGarithmic | LINear

Manual operation: See "Segment List" on page 489

CALCulate<Chn>:LIMit:SEGMent<Seg>:STIMulus:STARt <FreqPowTime>
CALCulate<Chn>:LIMit:SEGMent<Seg>:STIMulus:STOP <StimVal>

These commands change the start and stop stimulus values (i.e. the smallest and the largest stimulus values) of a limit line segment. A segment must be created first to enable the commands (e.g `CALCulate<Chn>:LIMit:DATA`). See also Section 4.4.1.1, "Rules for limit line definition", on page 136.

Tip: To define the stimulus values of several limit line segments with a single command, use `CALCulate<Chn>:LIMit:CONTrol[:DATA]`.

Suffix:
<Chn> Channel number used to identify the active trace

<Seg> Segment number

Parameters:
<StimVal> Stimulus value confining the limit line segment.
If specified, the unit of a stimulus value must be in accordance with the sweep type of the active channel (`[SENSe<Ch>:]SWEep:TYPE`). Default units are *Hz* for frequency sweeps, *dBm* for power sweeps, and *s* for time sweeps. For CW mode sweeps, stimulus values are dimensionless.

Default unit: NN

Example:
```
CALC:LIM:DATA 1,1500000000, 2000000000,2,3
```
Define an upper limit line segment (segment no. 1) in the stimulus range between 1.5 GHz and 2 GHz, assigning response values of +2 dB and +3 dB.
```
CALC:LIM:SEGM:STIM:STAR 1GHZ; STOP 2 GHZ; :
CALC:LIM:SEGM:TYPE LMIN
```
Change the segment to a lower limit line segment with a stimulus range between 1 GHz and 2 GHz.
```
CALC:LIM:DATA?
```
Query the type, the stimulus and response values of the created segment with a single command. The response is 2,1000000000,2000000000,2,3.

Manual operation: See "Segment List" on page 489

CALCulate<Chn>:LIMit:SEGMent<Seg>:TYPE <LimLineType>

Selects the limit line type for a limit line segment. This setting can be done before or after defining the stimulus and response values of the segment, however, a segment must be created first to enable this command (e.g `CALC:LIM:DATA`).

Note: The type command overwrites the `CALCulate<Chn>:LIMit:DATA` settings and is overwritten by them. It is not affected by the other commands in the `CALCulate<Chn>:LIMit...`, stimulus and response values of limit lines.

Suffix:
<Chn> Channel number used to identify the active trace

<Seg> Segment number

Parameters:
<LimLineType> LMIN | LMAX | OFF

Limit line type

Range: LMAX (upper limit line segment), LMIN (lower limit line segment), OFF (limit check switched off, limit line segment not deleted)

*RST: LMAX

Example: `*RST; :CALC:LIM:UPP 0, 0`
Define an upper limit line segment across the entire sweep range, using a constant upper limit of 0 dBm.
`CALC:LIM:SEGM:TYPE LMIN`
Turn the defined limit line segment into a lower limit line segment.

Manual operation: See "Segment List" on page 489

CALCulate<Chn>:LIMit:SOUNd[:STATe] <Boolean>

Deprecated command: Since version 3.45, generating sounds is no longer supported by the analyzer firmware.

Switches the acoustic signal (fail beep) on or off. The fail beep is generated each time the analyzer detects an exceeded limit.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<Boolean> ON | OFF - Fail beep on or off.

*RST: OFF

Example: `CALC:LIM:STAT ON; SOUN ON`
Switch on the limit check and activate the fail beep.

CALCulate<Chn>:LIMit:STATe <Boolean>

Switches the limit check (including upper and lower limits) on or off.

Tip: Use `CALCulate<Ch>:LIMit:UPPer:STATe` or `CALCulate<Ch>:LIMit:LOWer:STATe` to switch on or off the individual limit checks for upper or lower limit lines.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:

<Boolean> ON | OFF - Limit check on or off.
*RST: OFF

Example: `*RST; CALC:LIM:CONT 1 GHZ, 2 GHZ`
Define an upper limit line segment in the stimulus range between 1 GHz and 2 GHz, using default response values.
`CALC:LIM:STAT ON; FAIL?`
Switch on the limit check and query the result.

Manual operation: See "Limit Check" on page 485

CALCulate<Chn>:LIMit:STATe:AREA <HorizontalPos>, <VerticalPos>

Moves the limit check pass/fail message for the active trace <Chn> to one of nine predefined positions in the active diagram.

Suffix:

<Chn> Channel number used to identify the active trace

Parameters:

<HorizontalPos> LEFT | MID | RIGHt
Horizontal position

<VerticalPos> TOP | MID | BOTTom
Vertical position

Example: See `CALCulate<Chn>:LIMit:FAIL?`

Manual operation: See "Limit Check" on page 485

CALCulate<Chn>:LIMit:TTLout<Pt>[:STATe] <Boolean>

Switches the TTL pass/fail signals on or off. The signals are applied to the USER PORT as long as the active trace `<Chn>` is within limits, including the ripple limits.

See Section 13.3.1.1, "USER PORT", on page 1702.

Default values for new traces can be set using `SYSTem:TTLout<Pt>:STATus[:STATe]`.

Suffix:

<Chn> Channel number used to identify the active trace

<Pt> 1 - TTL out pass 1 (pin 13 of USER PORT connector)
2 - TTL out pass 2 (pin 14 of USER PORT connector

Parameters:

<Boolean> ON | OFF - TTL output signal on or off.
*RST: OFF

Example: `*RST; :CALC:LIM:CONT 1 GHZ, 2 GHZ`
Define an upper limit line segment in the stimulus range between 1 GHz and 2 GHz, using default response values.
`CALC:LIM:STAT ON; TTL2 ON`
Switch on the limit check and activate the TTL out pass 2 signal.

Manual operation: See "TTL1 Pass / TTL2 Pass" on page 487

CALCulate<Chn>:LIMit:X:OFFSet <Offset>
CALCulate<Chn>:LIMit:Y:OFFSet <Offset>

Shifts the stimulus/response values of all limit lines of channel <Chn>'s active trace, without modifying the constituent line segments.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<Offset> Default unit: NN

Manual operation: See "Shift Lines" on page 488

7.3.1.8 CALCulate:MARKer...

The `CALCulate:MARKer...` commands control the marker functions. The commands are device-specific and beyond what is specified in the SCPI subsystem `SOURce:MARKer...`.

Marker addressing

For all these commands, markers are addressed via:

1. Channel number <Chn>
   The command operates on the active trace of channel <Chn>.
2. Marker number <Mk>
   Selects a particular marker (ignored for reference markers).
   The command operates on marker <Mk> of channel <Chn>'s active trace.

CALCulate<Chn>:MARKer<Mk>:AOFF 1034
CALCulate<Chn>:MARKer<Mk>:BWIDth 1034
CALCulate:MARKer:COUPled[:STATe] 1035
CALCulate:MARKer:COUPled:TYPE 1036
CALCulate<Chn>:MARKer:DEFault:FORMat 1036
CALCulate<Chn>:MARKer<Mk>:DELTa[:STATe] 1038
CALCulate<Chn>:MARKer<Mk>:EXCursion 1038
CALCulate<Chn>:MARKer<Mk>:REFerence:EXCursion 1038
CALCulate<Chn>:MARKer<Mk>:EXCursion:STATe 1039
CALCulate<Chn>:MARKer<Mk>:REFerence:EXCursion:STATe 1039
CALCulate<Chn>:MARKer<Mk>:FORMat 1039
CALCulate<Chn>:MARKer<Mk>:REFerence:FORMat 1039

CALCulate:MARKer:FUNCtion:BWIDth:GMCenter 1040
CALCulate<Chn>:MARKer<Mk>:FUNCtion:BWIDth:MODE 1040
CALCulate<Chn>:MARKer<Mk>:FUNCtion:CENTer 1041
CALCulate<Chn>:MARKer<Mk>:REFerence:FUNCtion:CENTer 1041
CALCulate<Chn>:MARKer<Mk>:FUNCtion:DOMain:USER[:RANGe] 1041
CALCulate<Chn>:MARKer<Mk>:REFerence:FUNCtion:DOMain:USER[:RANGe] 1041
CALCulate<Chn>:MARKer<Mk>:FUNCtion:DOMain:USER:SHOW 1042
CALCulate<Chn>:MARKer<Mk>:REFerence:FUNCtion:DOMain:USER:SHOW 1042
CALCulate<Chn>:MARKer<Mk>:FUNCtion:DOMain:USER:STARt 1042
CALCulate<Chn>:MARKer<Mk>:FUNCtion:DOMain:USER:STOP 1042
CALCulate<Chn>:MARKer<Mk>:REFerence:FUNCtion:DOMain:USER:STARt 1042
CALCulate<Chn>:MARKer<Mk>:REFerence:FUNCtion:DOMain:USER:STOP 1042
CALCulate<Chn>:MARKer<Mk>:FUNCtion:EXECute 1043
CALCulate<Chn>:MARKer<Mk>:REFerence:FUNCtion:EXECute 1043
CALCulate<Chn>:MARKer<Mk>:FUNCtion:MAX 1044
CALCulate<Chn>:MARKer<Mk>:REFerence:FUNCtion:MAX 1044
CALCulate<Chn>:MARKer<Mk>:FUNCtion:MIN 1044
CALCulate<Chn>:MARKer<Mk>:REFerence:FUNCtion:MIN 1044
CALCulate<Chn>:MARKer<Mk>:FUNCtion:PEAK 1044
CALCulate<Chn>:MARKer<Mk>:FUNCtion:REFVal 1045
CALCulate<Chn>:MARKer<Mk>:REFerence:FUNCtion:REFVal 1045
CALCulate<Chn>:MARKer<Mk>:FUNCtion:RESult? 1045
CALCulate<Chn>:MARKer<Mk>:REFerence:FUNCtion:RESult? 1045
CALCulate<Chn>:MARKer<Mk>:FUNCtion:SETRef 1045
CALCulate<Chn>:MARKer<Mk>:FUNCtion:SPAN 1046
CALCulate<Chn>:MARKer<Mk>:FUNCtion:STARt 1046
CALCulate<Chn>:MARKer<Mk>:FUNCtion:STOP 1046
CALCulate<Chn>:MARKer<Mk>:REFerence:FUNCtion:STARt 1046
CALCulate<Chn>:MARKer<Mk>:REFerence:FUNCtion:STOP 1046
CALCulate<Chn>:MARKer<Mk>:FUNCtion:ZERodelay 1047
CALCulate<Chn>:MARKer<Mk>:REFerence:FUNCtion:ZERodelay 1047
CALCulate<Chn>:MARKer<Mk>:INFO 1047
CALCulate<Chn>:MARKer<Mk>:MODE 1047
CALCulate<Chn>:MARKer<Mk>:REFerence:MODE 1047
CALCulate<Chn>:MARKer:MPEak:EXCursion 1048
CALCulate<Chn>:MARKer:MPEak:EXCursion:STATe 1048
CALCulate<Chn>:MARKer:MPEak:THReshold 1049
CALCulate<Chn>:MARKer:MPEak:THReshold:STATe 1049
CALCulate<Chn>:MARKer<Mk>:NAME 1049
CALCulate<Chn>:MARKer<Mk>:REFerence:NAME 1049
CALCulate<Chn>:MARKer:SEARch:BFILter:RESult[:STATe] 1050
CALCulate<Chn>:MARKer:SEARch:BFILter:RESult[:STATe]:AREA 1050
CALCulate<Chn>:MARKer<Mk>:SEARch:FORMat 1050
CALCulate<Chn>:MARKer<Mk>:REFerence:SEARch:FORMat 1050
CALCulate<Chn>:MARKer<Mk>:SEARch:TRACking 1052
CALCulate<Chn>:MARKer<Mk>:REFerence:SEARch:TRACking 1052
CALCulate<Chn>:MARKer<Mk>[:STATe] 1053
CALCulate<Chn>:MARKer<Mk>:REFerence[:STATe] 1053
CALCulate<Chn>:MARKer[:STATe]:AREA 1053
CALCulate<Chn>:MARKer<Mk>:STYLe 1053

CALCulate<Chn>:MARKer<Mk>:TARGet 1054
CALCulate<Chn>:MARKer<Mk>:REFerence:TARGet 1054
CALCulate<Chn>:MARKer<Mk>:THReshold 1055
CALCulate<Chn>:MARKer<Mk>:REFerence:THReshold 1055
CALCulate<Chn>:MARKer<Mk>:THReshold:STATe 1055
CALCulate<Chn>:MARKer<Mk>:REFerence:THReshold:STATe 1055
CALCulate<Chn>:MARKer<Mk>:TYPE 1056
CALCulate<Chn>:MARKer<Mk>:REFerence:TYPE 1056
CALCulate<Chn>:MARKer<Mk>:X 1057
CALCulate<Chn>:MARKer<Mk>:REFerence:X 1057
CALCulate<Chn>:MARKer<Mk>:Y 1058
CALCulate<Chn>:MARKer<Mk>:REFerence:Y 1058

CALCulate<Chn>:MARKer<Mk>:AOFF

Removes all markers from all traces of the active recall set. The removed markers remember their properties (stimulus value, format, delta mode, number) when they are restored (`CALCulate<Chn>:MARKer<Mk>[:STATe] ON`). The marker properties are definitely lost if the associated trace is deleted.

Suffix:

<Chn> Channel number used to identify the active trace. If unspecified the numeric suffix is set to 1.

<Mk> This numeric suffix is ignored and can be set to any value.

Example: Suppose that the active recall set contains an active trace no. 1.
`CALC:MARK1 ON; MARK2 ON`
Create markers 1 and 2 and assign them to the trace no. 1.
`CALC:MARK:AOFF`
Remove both markers.

Usage: Event

Manual operation: See "All Off" on page 504

CALCulate<Chn>:MARKer<Mk>:BWIDth <Bandwidth>

Sets the bandfilter level for a bandfilter search or returns the results. The command is only available after a bandfilter search has been executed (`CALCulate<Chn>:MARKer<Mk>:FUNCtion:EXECute BFILter`; see example below).

The response to the query `CALCulate<Chn>:MARKer<Mk>:BWIDth?` contains the following bandfilter search results:

- <Bandwidth> – bandwidth of the bandpass/bandstop region.
- <Center> – stimulus frequency at the center of the bandpass/bandstop region (the stimulus value of marker M4).
- <Quality Factor (3 dB)> – quality factor, i.e. the ratio between the center frequency and the 3-dB bandwidth.
- <Loss> – loss at the center of the bandpass/bandstop region (the response value of marker M4 at the time of the bandfilter search).

- <LowerEdge> – lower band edge.
- <UpperEdge> – upper band edge.

Tip: To obtain the <Quality Factor (BW)> result from the bandfilter info field, calculate the ratio <Center> / <Bandwidth>.

Suffix:

<Chn> Channel number used to identify the active trace

<Mk> This numeric suffix is ignored and can be set to any value because the bandfilter search functions always use markers M1 to M4.

Parameters:

<Bandwidth> Difference between the band edges and the center response value of a bandfilter peak; must be negative for a bandpass search and positive for a bandstop search.

Range: For bandpass: -100.00 dB to -0.01 dB; for bandstop: +0.01 dB to +100.00 dB
Increment: 0.03 dB
*RST: -3 dB
Default unit: dB

Example:

```
CALC:MARK:FUNC:BWID:MODE BST
```
Select a bandstop filter search.
```
CALC:MARK:FUNC:EXEC BFIL
```
Initiate the bandpass filter search for the current trace. Create markers M1 to M4.
```
CALC:MARK:SEAR:BFIL:RES ON
```
Display the marker info field in the diagram area.
```
CALC:MARK:BWID 6
```
Select a 6-dB bandwidth for the bandstop.
```
CALC:MARK:BWID?
```
Query the results of the bandfilter search. An error message is generated if the bandfilter search fails so that no valid results are available.
```
CALC:MARK:SEAR:BFIL:RES:AREA LEFT, TOP
```
For a subsequent check at the GUI or a hardcopy, move the info field to the top-left position.

Manual operation: See "Bandwidth" on page 519

CALCulate:MARKer:COUPled[:STATe] <Boolean>

Enables marker coupling to the active trace of the active channel or disables it.

Use `CALCulate:MARKer:COUPled:TYPE` to select the suitable coupling type before setting `CALCulate:MARKer:COUPled:STATe` to `ON`.

Parameters:

<Boolean> ON | OFF - enables or disables marker coupling.

*RST: OFF

Example: Suppose that the active recall set contains traces Trc1 and Trc2, assigned to channels no. 1 and 2, respectively.
`:CALC2:PAR:SEL 'TRC2'; :CALC2:MARK1 ON; MARK2 ON`
Select Trc2 as the active trace of channel 2 and activate markers 1 and 2 for it. The default position for both markers is the center of the sweep range.
`INSTrument:NSELect 2`
Make channel 2 the active channel.
`CALCulate:MARKer:COUPled:TYPE ALL`
Select coupling type "All" (recall set-wide coupling).
`CALC:MARK:COUP ON`
Activate marker coupling to Trc2 (the active trace of the active channel). Automatically creates markers no. 1 and 2 on Trc1 and couples them to the respective markers of Trc2.

Manual operation: See "Coupled Markers" on page 506

CALCulate:MARKer:COUPled:TYPE <Type>

Selects the marker coupling type to be used in the current recall set.

Marker coupling is enabled using `CALCulate:MARKer:COUPled[:STATe] ON`. Changing the coupling type automatically disables marker coupling throughout the recall set.

Parameters:

<Type> ALL | CHANnel | DIAGram

Marker coupling mode

ALL
recall set-wide

CHANnel
channel-wide

DIAGram
diagram-wide

Example: See `CALCulate:MARKer:COUPled[:STATe]` on page 1035

Manual operation: See "Coupling Type" on page 525

CALCulate<Chn>:MARKer:DEFault:FORMat <OutFormat>

Defines the default marker format of the selected channel's active trace.

New markers are formatted with the default marker format; previously existing markers will be reformatted if (and only if) their marker format is set to (Trace) `DEFault` (using `CALCulate<Chn>:MARKer<Mk>:FORMat`).

Suffix:

<Chn> Channel number used to identify the active trace

Parameters:

<OutFormat> MLINear | MLOGarithmic | PHASe | POLar | GDELay | REAL | IMAGinary | SWR | LINPhase | LOGPhase | IMPedance | ADMittance | DEFault | COMPlex | MDB | MLPHase | MDPHase | MIMPedance | PIMPedance | PADMittance | MADMittance | MPIMpedance | MPADmittance | INDX | GDDerivation | NOISe

DEFault means the default marker format is dynamically adjusted to the selected trace format (`CALCulate<Chn>:FORMat`). For the other marker formats see the table below and the description in "Marker format" on page 99.

*RST: DEFault

Manual operation: See "Dflt Marker Frmt" on page 443

SCPI	GUI
MLINear	"Lin Mag"
MLOGarithmic MDB (for R&S ZVR compatibility)	"dB Mag"
PHASe	"Phase"
POLar COMPlex (for R&S ZVR compatibility)	"Real Imag"
GDELay	"Delay"
GDDerivation	"Delay Derivation"
REAL	"Real"
IMAGinary	"Imag"
SWR	"SWR"
LINPhase MLPhase (for R&S ZVR compatibility)	"Lin Mag Phas"e
LOGPhase MDPhase (for R&S ZVR compatibility)	"dB Mag Phase"
IMPedance	"R + j X" for reflection measurement traces "R + j X series" for transmission measurement traces
PIMPedance	"R + j X", for reflection measurement traces "R + j X parallel" for transmission measurement traces
ADMittance	"G + j B" for reflection measurement traces "G + j B series" for transmission measurement traces
PADMittance	"G + j B" for reflection measurement traces "G + j B parallel" for transmission measurement traces

SCPI	GUI
MIMPedance	"Imp Mag" for reflection measurement traces "Imp Mag series" for transmission measurement traces
MPIMpedance	"Imp Mag" for reflection measurement traces "Imp Mag parallel" for transmission measurement traces
MADMittance	"Adm Mag" for reflection measurement traces "Adm Mag series" for transmission measurement traces
MPADmittance	"Adm Mag" for reflection measurement traces "Adm Mag parallel" for transmission measurement traces
INDex	Index
NOISe	Noise

CALCulate<Chn>:MARKer<Mk>:DELTa[:STATe] <Boolean>

Switches the delta mode for marker `<Mk>` on trace no. `<Chn>` on or off. The marker must be created before using `CALCulate<Chn>:MARKer<Mk>[:STATe] ON`. If the active trace contains no reference marker, the command also creates a reference marker.

Suffix:

<Chn> Channel number used to identify the active trace

<Mk> Marker number.

Parameters:

<Boolean> ON | OFF - Enables or disables the delta mode.

*RST: OFF

Example: Suppose that the active recall set contains an active trace no. 1.

```
CALC:MARK ON
```

Create marker no. 1 and set it to the center of the sweep range.

```
CALC:MARK:DELT ON
```

Create a reference marker at the center of the sweep range and set marker 1 to delta mode.

Manual operation: See "Delta Mode" on page 505

CALCulate<Chn>:MARKer<Mk>:EXCursion <Value>
CALCulate<Chn>:MARKer<Mk>:REFerence:EXCursion <TargetSearchVal>

Defines a marker-specific minimum excursion value for peak searches.

Use `CALCulate<Chn>:MARKer<Mk>:EXCursion:STATe` or `CALCulate<Chn>:MARKer<Mk>:REFerence:EXCursion:STATe` to activate it.

Suffix:
<Chn> Channel number used to identify the active trace

<Mk> Marker number (ignored for reference markers)

Parameters:
<Value> Minimum peak excursion
The unit is derived from the active trace format and cannot be modified. Changing the trace format resets the excursion to a format-specific default value.

Manual operation: See "Excursion Settings" on page 513

CALCulate<Chn>:MARKer<Mk>:EXCursion:STATe <Boolean>
CALCulate<Chn>:MARKer<Mk>:REFerence:EXCursion:STATe <Boolean>

Activates or deactivates the marker-specific minimum excursion for peak searches.

Use `CALCulate<Chn>:MARKer<Mk>:EXCursion` or `CALCulate<Chn>:MARKer<Mk>:REFerence:EXCursion` to set the minimum peak excursion.

Suffix:
<Chn> Channel number used to identify the active trace

<Mk> Marker number (ignored for reference markers)

Parameters:
<Active> Boolean

Manual operation: See "Excursion Settings" on page 513

CALCulate<Chn>:MARKer<Mk>:FORMat <OutFormat>
CALCulate<Chn>:MARKer<Mk>:REFerence:FORMat <OutFormat>

Sets/queries the output format for the (complex) value of the related marker.

Suffix:
<Chn> Channel number used to identify the active trace

<Mk> Marker number (ignored for reference markers)

Parameters:

<OutFormat> MLINear | MLOGarithmic | PHASe | POLar | GDELay | REAL | IMAGinary | SWR | LINPhase | LOGPhase | IMPedance | ADMittance | DEFault | COMPlex | MDB | MLPHase | MDPHase | MIMPedance | PIMPedance | PADMittance | MADMittance | MPIMpedance | MPADmittance | INDX | GDDerivation | NOISe

`DEFault` means that the reference marker is formatted according to the related trace's default marker format (see `CALCulate<Chn>:MARKer:DEFault:FORMat`).
For the other marker formats, see `CALCulate<Chn>:MARKer:DEFault:FORMat` and the description in "Marker format" on page 99.

*RST: DEFault

Manual operation: See "Marker Format" on page 507

CALCulate:MARKer:FUNCtion:BWIDth:GMCenter <arg0>

Defines how bandfilter searches calculate the center frequency of the passband or stopband.

Parameters:

<arg0> **ON** – use **geometric** mean of lower and upper band edge
OFF – use **arithmetic** mean

*RST: n/a (*RST does not affect the calculation rule; the factory setting is ON/geometric mean)

Manual operation: See "Geometric Calculation of Bandfilter Center" on page 818

CALCulate<Chn>:MARKer<Mk>:FUNCtion:BWIDth:MODE <BandfilterType>

Selects the bandfilter search mode. In contrast to manual control, bandfilter tracking is not automatically activated.

Suffix:

<Chn> Channel number used to identify the active trace

<Mk> This numeric suffix is ignored and can be set to any value because the bandfilter search functions always use markers M1 to M4.

Parameters:

<BandfilterType> BPASs | BSTop | BPRMarker | BSRMarker | BPABsolute | BSABsolute | NONE

Bandfilter search type:
BPASs – Bandpass Search Ref to Max
BSTop – Bandstop Search Ref to Max
BPRMarker – Bandpass Search Ref to Marker
BSRMarker – Bandstop Search Ref to Marker
BPABsolute – Bandpass Absolute Level

BSABsolute – Bandstop Absolute Level
NONE – deactivate bandfilter search, result off

*RST: NONE

Example: See `CALCulate<Chn>:MARKer<Mk>:BWIDth`

Manual operation: See "Bandpass Ref to Max" on page 519

CALCulate<Chn>:MARKer<Mk>:FUNCtion:CENTer
CALCulate<Chn>:MARKer<Mk>:REFerence:FUNCtion:CENTer

Sets the center of the sweep range to the stimulus value of the related marker (marker <Mk> or the reference marker on the active trace of channel `<Chn>`).

Suffix:
<Chn> Channel number used to identify the active trace

<Mk> Marker number (ignored for reference markers)

Example:
```
*RST; :CALC:MARK ON
```
Create marker 1 in the center of the current sweep range and assign it to trace no. 1.
```
CALC:MARK:FUNC:CENT
```
Leave the sweep range unchanged.

Usage: Event

Manual operation: See "Center = Marker / Start = Marker / Stop = Marker / Span = Marker" on page 523

CALCulate<Chn>:MARKer<Mk>:FUNCtion:DOMain:USER[:RANGe]
<NumSearchRange>
CALCulate<Chn>:MARKer<Mk>:REFerence:FUNCtion:DOMain:USER[:RANGe]
<NumSearchRange>

Assigns search range no. `<NumSearchRange>` to the related marker (marker `<Mk>` or the reference marker of channel <Chn>'s active trace).

Selects the search range, e.g. to display range limit lines or define the start and stop values.

Suffix:
<Chn> Channel number used to identify the active trace

<Mk> Marker number (ignored for reference markers)

Parameters:
<NumSearchRange> Number of the search range.

Range: 0 to 10 where 0 refers to the fixed full span search range (equal to the sweep range) and 1 to 10 refer to user-definable search ranges; see example.

*RST: 0 (reserved for full span search range)

Example: CALC1:MARK1:FUNC:DOM:USER 2
Select the search range no. 2, assigned to marker no. 1 and trace no. 1.
CALC:MARK:FUNC:DOM:USER:STARt 1GHz
Set the start frequency of the search range to 1 GHz.
CALC:MARK:FUNC:DOM:USER:STOP 1.2GHz
Set the stop frequency of the search range to 1.2 GHz.

Manual operation: See "Search Range" on page 511

CALCulate<Chn>:MARKer<Mk>:FUNCtion:DOMain:USER:SHOW <Boolean>
CALCulate<Chn>:MARKer<Mk>:REFerence:FUNCtion:DOMain:USER:SHOW <Boolean>

Displays or hides range limit lines for the search range selected via CALCulate<Chn>:MARKer<Mk>:FUNCtion:DOMain:USER[:RANGe] or CALCulate<Chn>:MARKer<Mk>:REFerence:FUNCtion:DOMain:USER[:RANGe].

Suffix:
<Chn> Channel number used to identify the active trace
<Mk> Marker number (ignored for reference markers)

Parameters:
<Boolean> ON | OFF - range limit lines on or off.
*RST: OFF

Example: See CALCulate<Chn>:STATistics:DOMain:USER

Manual operation: See "Range Limit Lines On" on page 512

CALCulate<Chn>:MARKer<Mk>:FUNCtion:DOMain:USER:STARt <StarSearchRange>
CALCulate<Chn>:MARKer<Mk>:FUNCtion:DOMain:USER:STOP <StopSearchRange>
CALCulate<Chn>:MARKer<Mk>:REFerence:FUNCtion:DOMain:USER:STARt <StarSearchRange>
CALCulate<Chn>:MARKer<Mk>:REFerence:FUNCtion:DOMain:USER:STOP <Value>

These commands define the start and stop values of the search range selected via CALCulate<Chn>:MARKer<Mk>:FUNCtion:DOMain:USER[:RANGe] or CALCulate<Chn>:MARKer<Mk>:REFerence:FUNCtion:DOMain:USER[:RANGe].

Suffix:
<Chn> Channel number used to identify the active trace
<Mk> Marker number (ignored for reference markers)

Parameters:

<Value> Beginning or end of the search range.

Range: Maximum allowed sweep range, depending on the instrument model and on the sweep type.

Default unit: NN

Example: See `CALCulate<Chn>:MARKer<Mk>:FUNCtion:DOMain:USER[:RANGe]`

Manual operation: See "Search Range" on page 511

CALCulate<Chn>:MARKer<Mk>:FUNCtion:EXECute [<SearchMode>]
CALCulate<Chn>:MARKer<Mk>:REFerence:FUNCtion:EXECute [<SearchMode>]

Selects a search mode for the related marker (see "Marker addressing" on page 1032) and executes the search.

Except for bandfilter searches (`BFILter`), you have to create the related marker using `CALCulate<Chn>:MARKer<Mk>[:STATe] ON` or `CALCulate<Chn>:MARKer<Mk>:REFerence[:STATe] ON` before using this command.

Suffix:

<Chn> Channel number used to identify the active trace

<Mk> Marker number
Ignored for reference markers and for bandfilter searches (`BFILter`), which always use markers M1 to M4.

Setting parameters:

<SearchMode> NONE | MAXimum | MINimum | RPEak | LPEak | NPEak | TARGet | LTARget | RTARget | BFILter | MMAXimum | MMINimum | SPRogress

See list of search modes below.

Example: Suppose that the active recall set contains an active trace no. 1.
`CALC:MARK ON`
Create marker M1 and assign it to trace no. 1.
`CALC:MARK:FUNC:EXEC MAX; RES?`
Move the created marker to the absolute maximum of the trace and query the stimulus and response value of the search result.

Usage: Setting only

Manual operation: See "Max / Min" on page 509

The analyzer provides the following search modes:

Mode	Find...
MAXimum	Absolute maximum in the search range (see `CALCulate<Chn>:MARKer<Mk>:FUNCtion:DOMain:USER[:RANGe]`)
MINimum	Absolute maximum in the search range
RPEak	Next valid peak to the right of the current marker position

Mode	Find...
LPEak	Next valid peak to the left
NPEak	Next highest or lowest value among the valid peaks (next peak)
TARGet	Target value (see CALCulate<Chn>:MARKer<Mk>:TARGet)
RTARget	Next target value to the right of the current marker position
LTARget	Next target value to the left
BFILter Regular markers only	Bandfilter search. The results are queried using CALCulate<Chn>:MARKer<Mk>:BWIDth.
MMAXimum or MMINimum	Multiple peak search
SPRogress	Sweep progress

For RPEak, LPEak and NPEak, use CALCulate<Chn>:MARKer<Mk>:FUNCtion:PEAK on page 1044 to select the peak type to be searched for.

CALCulate<Chn>:MARKer<Mk>:FUNCtion:MAX
CALCulate<Chn>:MARKer<Mk>:REFerence:FUNCtion:MAX
CALCulate<Chn>:MARKer<Mk>:FUNCtion:MIN
CALCulate<Chn>:MARKer<Mk>:REFerence:FUNCtion:MIN

Sets the upper/lower edge of the diagram equal to the response value of the active marker, leaving the values of the vertical divisions ("Scale / Div") unchanged. Upper for MAX / lower for MIN.

Suffix:
<Chn> Channel number

<Mk> Marker number (ignored for reference markers)

Usage: Event

Manual operation: See "Ref Val = Marker / Max = Marker / Min = Marker" on page 523

CALCulate<Chn>:MARKer<Mk>:FUNCtion:PEAK [<PeakMode>]

Defines the peak type to be searched for using CALCulate<Chn>:MARKer<Mk>[:REFerence]:FUNCtion:EXECute RPEak | LPEak | NPEak.

Suffix:
<Chn> Channel number used to identify the active trace

<Mk> Marker number

Setting parameters:
<PeakMode> MAXimum | MINimum | MINMaximum

Usage: Setting only

Manual operation: See "Peak Type" on page 510

CALCulate<Chn>:MARKer<Mk>:FUNCtion:REFVal
CALCulate<Chn>:MARKer<Mk>:REFerence:FUNCtion:REFVal

Sets the reference value equal to the response value of the active marker, leaving the values of the vertical divisions ("Scale / Div") unchanged.

Suffix:
<Chn> Channel number

<Mk> Marker number (ignored for reference markers)

Usage: Event

Manual operation: See "Ref Val = Marker / Max = Marker / Min = Marker" on page 523

CALCulate<Chn>:MARKer<Mk>:FUNCtion:RESult?
CALCulate<Chn>:MARKer<Mk>:REFerence:FUNCtion:RESult?

Returns the result (stimulus and response value) of a search started using `CALCulate<Chn>:MARKer<Mk>:FUNCtion:EXECute` or `CALCulate<Chn>:MARKer<Mk>:REFerence:FUNCtion:EXECute`. The search must be executed before the command is enabled.

Suffix:
<Chn> Channel number used to identify the active trace

<Mk> Marker number (ignored)

Example: See `CALCulate<Chn>:MARKer<Mk>:FUNCtion:EXECute`

Usage: Query only

Manual operation: See "Max / Min" on page 509

CALCulate<Chn>:MARKer<Mk>:FUNCtion:SETRef

Places the reference marker to the position of the marker with number <Mk>.

Suffix:
<Chn> Channel number

<Mk> Marker number

Usage: Event

Manual operation: See "Ref Mkr -> Mkr" on page 508

CALCulate<Chn>:MARKer<Mk>:FUNCtion:SPAN

Sets the span of the sweep range equal to the absolute value of the first coordinate of the active delta marker <Mk> on trace no. <Chn>.

Suffix:

<Chn> Channel number used to identify the active trace

<Mk> Marker number.

Example:

```
*RST; :CALC:MARK ON; MARK:DELTa ON
```

Create marker 1 in the center of the current sweep range and enable the delta mode.

```
CALC:MARK:X 300MHz
```

Increase the stimulus value of the delta marker by 300 MHz.

```
CALC:MARK:FUNC:SPAN
```

Set the sweep range equal to 300 MHz. The sweep range starts at the reference marker position, i.e. in the center of the analyzer's frequency range.

Usage: Event

Manual operation: See "Center = Marker / Start = Marker / Stop = Marker / Span = Marker" on page 523

CALCulate<Chn>:MARKer<Mk>:FUNCtion:STARt
CALCulate<Chn>:MARKer<Mk>:FUNCtion:STOP
CALCulate<Chn>:MARKer<Mk>:REFerence:FUNCtion:STARt
CALCulate<Chn>:MARKer<Mk>:REFerence:FUNCtion:STOP

These commands set the STARt or STOP of the sweep range to the stimulus value of the related marker (see "Marker addressing" on page 1032).

Suffix:

<Chn> Channel number used to identify the active trace

<Mk> Marker number (ignored for reference markers)

Example:

```
*RST; :CALC:MARK ON
```

Create marker 1 in the center of the current sweep range and assign it to trace no. 1.

```
CALC:MARK:FUNC:STAR
```

Divide the sweep range in half, starting at the current marker position. As an alternative:

```
CALC:MARK:FUNC:STOP
```

Divide the sweep range in half, ending at the current marker position.

Usage: Event

Manual operation: See "Center = Marker / Start = Marker / Stop = Marker / Span = Marker" on page 523

CALCulate<Chn>:MARKer<Mk>:FUNCtion:ZERodelay
CALCulate<Chn>:MARKer<Mk>:REFerence:FUNCtion:ZERodelay

Shifts the trace in vertical direction so that the delay at the marker position becomes zero. Mathematically, it modifies the measurement results by subtracting the delay at the current marker position. Only available for Delay traces.

Suffix:
<Chn> Channel number

<Mk> Marker number (ignored for reference markers)

Usage: Event

Manual operation: See "Zero Delay at Marker" on page 523

CALCulate<Chn>:MARKer<Mk>:INFO <Boolean>

Defines the visibility of the dedicated marker info field for channel no. <Chn>'s marker no. <Mk>.

This info field is shown at the corresponding marker symbol. You can define its content using `CALCulate<Chn>:MARKer<Mk>:FORMat`

Suffix:
<Chn> Channel number used to identify the active trace

<Mk> Marker number

Parameters:
<Boolean>

OFF (0)
No dedicated marker info

ON (1)
Dedicated marker info visible (if the marker itself is visible)

*RST: OFF (0)

Manual operation: See "Marker Info" on page 508

CALCulate<Chn>:MARKer<Mk>:MODE <Mode>
CALCulate<Chn>:MARKer<Mk>:REFerence:MODE <Mode>

Defines the positioning mode of the related marker (see "Marker addressing" on page 1032). You don't have to create the marker before, the mode can be assigned in advance.

Suffix:
<Chn> Channel number used to identify the active trace

<Mk> Marker number (ignored for reference markers)

Parameters:

<Mode> CONTinuous | DISCrete

CONTinuous –- marker can be positioned on any point of the trace, and its response values are obtained by interpolation.
DISCrete – marker can be set to discrete sweep points only.

*RST: CONT

Example: Suppose that the active recall set contains an active trace no. 1.

```
CALC:MARK:MODE DISC; :CALC:MARK2:MODE CONT
```

Create marker 1 in discrete mode and marker 2 in continuous mode.

```
CALC:MARK ON; MARK2 ON
```

Display the two markers. Due to the different modes, the horizontal positions can be different.

Example: Suppose that the active recall set contains an active trace no. 1.

```
CALC:MARK:REF:MODE DISC
CALC:MARK2:REF:MODE CONT
```

Create the reference marker in discrete mode and marker 2 in continuous mode.

```
CALC:MARK:REF ON; :CALC:MARK2 ON
```

Display the two markers. Due to the different modes, the horizontal positions can be different.

Manual operation: See "Discrete" on page 507

CALCulate<Chn>:MARKer:MPEak:EXCursion <TargetSearchVal>

Defines a minimum excursion value for multiple peak searches.

Use `CALCulate<Chn>:MARKer:MPEak:EXCursion:STATe` to activate it.

Suffix:

<Chn> Channel number used to identify the active trace

Parameters:

<Value> Minimum peak excursion
The unit is derived from the active trace format and cannot be modified. Changing the trace format resets the excursion to a format-specific default value.

Manual operation: See "Excursion Settings" on page 516

CALCulate<Chn>:MARKer:MPEak:EXCursion:STATe <Boolean>

Activates or deactivates the minimum excursion for multiple peak searches.

Use `CALCulate<Chn>:MARKer:MPEak:EXCursion` to set the minimum peak excursion.

Suffix:

<Chn> Channel number used to identify the active trace

Parameters:
<Active> Boolean

Manual operation: See "Excursion Settings" on page 516

CALCulate<Chn>:MARKer:MPEak:THReshold <Value>

Defines a threshold value for multiple peak searches.

Use `CALCulate<Chn>:MARKer:MPEak:THReshold:STATe` to activate it.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<Value> Threshold value
The unit is derived from the active trace format and cannot be modified. Changing the trace format resets the threshold to a format-specific default value.

Manual operation: See "Threshold Settings" on page 515

CALCulate<Chn>:MARKer:MPEak:THReshold:STATe <Boolean>

Activates or deactivates the threshold for multiple peak searches.

Use `CALCulate<Chn>:MARKer:MPEak:THReshold` to set the threshold value.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<Active> Boolean

Manual operation: See "Threshold Settings" on page 515

CALCulate<Chn>:MARKer<Mk>:NAME <MarkerName>
CALCulate<Chn>:MARKer<Mk>:REFerence:NAME <MarkerName>

Defines a name for the related marker (see "Marker addressing" on page 1032). You do not have to create the marker before, the name can be assigned in advance.

Suffix:
<Chn> Channel number used to identify the active trace

<Mk> Marker number (ignored for reference markers)

Parameters:
<MarkerName> Marker name (string parameter)
*RST: 'M1' for marker no. 1 etc.

Example: Suppose that the active recall set contains an active trace no. 1.
```
CALC:MARK:NAME '&$% 1'; :CALC:MARK ON
```
Create marker 1, name it "&$% 1", and display it.
```
CALC:MARK:REF ON
CALC:MARK:REF:NAME 'Reference'
```
Display the reference marker and rename it "Reference".

Manual operation: See "Marker Name" on page 506

CALCulate<Chn>:MARKer:SEARch:BFILter:RESult[:STATe] <Boolean>

Shows or hides the bandfilter search results in the diagram area.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<Boolean> ON - show the bandfilter search results. If no bandfilter search has been initiated before (`CALCulate<Chn>:MARKer<Mk>:FUNCtion:EXECute BFILter`), nothing is displayed.
OFF - hide the bandfilter search results.
*RST: OFF

Example: See `CALCulate<Chn>:MARKer<Mk>:BWIDth`

Manual operation: See "Result Off" on page 521

CALCulate<Chn>:MARKer:SEARch:BFILter:RESult[:STATe]:AREA
<HorizontalPos>, <VerticalPos>

Moves the bandfilter search info field for the active trace `<Chn>` to one of nine predefined positions in the active diagram.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<HorizontalPos> LEFT | MID | RIGHt
Horizontal position

<VerticalPos> TOP | MID | BOTTom
Vertical position

Example: See `CALCulate<Chn>:MARKer<Mk>:BWIDth`

Manual operation: See "Bandpass Ref to Max" on page 519

CALCulate<Chn>:MARKer<Mk>:SEARch:FORMat <SearchFormat>
CALCulate<Chn>:MARKer<Mk>:REFerence:SEARch:FORMat <SearchFormat>

Defines the format in which you want to specify the target value `CALCulate<Chn>:MARKer<Mk>:TARGet` for the related marker (see "Marker addressing" on page 1032).

Each marker can have a different target format. The table below gives an overview on how a complex target value z = x + jy is converted.

Target Format	Description	Formula
MLINear	Magnitude of z, unconverted.	$\|z\| = \sqrt{(x^2 + y^2)}$
MLOGarithmic	Magnitude of z in dB	Mag(z) = 20 log|z| dB
PHASe	Phase of z	φ (z) = arctan (y/x)
UPHase	Unwrapped phase of z comprising the complete number of 360° phase rotations	Φ(z) = φ (z) + 2k·360°
REAL	Real part of z	Re(z) = x
IMAGinary	Imaginary part of z	Im(z) = y
SWR	(Voltage) Standing Wave Ratio	SWR = (1 + |z|) / (1 – |z|)
DEFault	Identical to trace format. **Note**: the Smith and Polar traces use "Lin Mag" as the default format for the target value.	-

Suffix:

<Chn> Channel number used to identify the active trace

<Mk> Marker number (ignored for reference markers)

Parameters:

<SearchFormat> MLINear | MLOGarithmic | PHASe | UPHase | REAL | IMAGinary | SWR | DEFault

The format of the target value in the marker target search (`CALCulate<Chn>:MARKer<Mk>:FUNCtion:EXECute TARGet` or `CALCulate<Chn>:MARKer<Mk>:REFerence:FUNCtion:EXECute TARGet`). See table above.

*RST: DEFault

Example: Suppose channel 1's selected trace is POLar and marker 1 was not created yet.
```
:CALCULATE1:MARKER1 ON
```
Create/enable Marker 1
```
:CALCulate1:MARKer1:FUNCtion:SELect TARGet
```
Select TARGet search mode for marker 1.
```
:CALCulate1:MARKer1:SEARch:FORMat?
```
Query the target format of marker 1. The result is DEF, and for polar diagrams, the default target format is PHASe.
```
:CALCulate1:MARKer1:FUNCtion:TARGet?
```
Query for the default target value. For PHASe, it is 0 (degrees).
```
:CALCulate1:MARKer1:SEARch:FORMat MLOGarithmic
```
Change the target search format to dB magnitude.
```
:CALCulate1:MARKer1:FUNCtion:TARGet?
```
Query for the default target value. For dB magnitude, it is 0 (dB).
```
:CALCulate1:MARKer1:FUNCtion:TARGet -3
```
Set the target value to -3 dB.
```
:CALCulate1:MARKer1:FUNCtion:EXECute
```
Execute the target search for marker 1.
```
:CALCulate1:MARKer1:FUNCtion:RESult?
```
Query for the results.

Manual operation: See "Target Format" on page 517

CALCulate<Chn>:MARKer<Mk>:SEARch:TRACking <Boolean>
CALCulate<Chn>:MARKer<Mk>:REFerence:SEARch:TRACking <Boolean>

These commands enable or disable tracking for the respective marker (see "Marker addressing" on page 1032).

They are only available if a search mode is active for the respective marker (see `CALCulate<Chn>:MARKer<Mk>:FUNCtion:EXECute <SearchMode>` or `CALCulate<Chn>:MARKer<Mk>:REFerence:FUNCtion:EXECute <SearchMode>`), in which case tracking causes the search logic to be repeated after each sweep.

Tip: If the current search mode is a bandfilter or multiple peak search (regular markers only), this command enables or disables the corresponding tracking.

Suffix:

<Chn> Channel number used to identify the active trace

<Mk> Marker number
Ignored not only for reference markers, but also for bandfilter and multiple peak searches, where tracking recalculates the whole marker set for each sweep.

Parameters:

<Boolean> ON | OFF - enables or disables the marker tracking mode.
*RST: OFF

Example: Suppose the active recall set contains an active trace no. 1.
`CALC:MARK ON; :CALC:MARK:FUNC:EXEC MAXimum`
Create marker no. 1 and assign it to trace no. 1. Activate a maximum search for marker no. 1.
`CALC:MARK:SEAR:TRAC ON`
Enable the tracking mode for the created marker.

Manual operation: See "Tracking" on page 510

CALCulate<Chn>:MARKer<Mk>[:STATe] <Boolean>
CALCulate<Chn>:MARKer<Mk>:REFerence[:STATe] <Boolean>

Creates or removes the related marker (see "Marker addressing" on page 1032).

Suffix:
<Chn> Channel number used to identify the active trace

<Mk> Marker number (ignored for reference markers)

Parameters:
<Boolean> ON | OFF – creates or removes the marker.

*RST: OFF

Example: Suppose that the active recall set contains an active trace no. 1.
`CALC:MARK ON; MARK2 ON`
Create markers 1 and 2 and assign them to trace no. 1. The default position of both markers is the center of the sweep range.

Manual operation: See "On" on page 504

CALCulate<Chn>:MARKer[:STATe]:AREA <HorizontalPos>, <VerticalPos>

Moves the marker info field for the active trace `<Chn>` to one of nine predefined positions in the active diagram.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<HorizontalPos> LEFT | MID | RIGHt

Horizontal position

<VerticalPos> TOP | MID | BOTTom

Vertical position

Example: See `CALCulate<Chn>:MARKer<Mk>:Y`

Manual operation: See "Mkr 1 ... Mkr 10" on page 505

CALCulate<Chn>:MARKer<Mk>:STYLe <Style>

Defines the graphical representation of channel no. <Chn>'s marker no. <Mk>.

Suffix:
<Chn> Channel number used to identify the active trace

<Mk> Marker number

Parameters:
<Style> TRIangle | VLINe | HLINe | CROSsbar | HTRiangle | VMARk | CMARk

TRIangle
Filled triangle

VLINe
Vertical line

HLINe
Horizontal line

CROSsbar
Crossbar (vertical + horizontal line)

HTRiangle
Hollow triangle

VMARk
Vertical mark (short vertical line)

CMARk
Cross mark

Manual operation: See "Marker Style" on page 507

CALCulate<Chn>:MARKer<Mk>:TARGet <TargetSearchVal>
CALCulate<Chn>:MARKer<Mk>:REFerence:TARGet <TargetSearchVal>

Defines the target value for the target search of marker the related marker (see "Marker addressing" on page 1032).

The target search can be activated using `CALCulate<Chn>:MARKer<Mk>:FUNCtion:EXECute TARGet` or `CALCulate<Chn>:MARKer<Mk>:REFerence:FUNCtion:EXECute TARGet`.

Suffix:
<Chn> Channel number used to identify the active trace

<Mk> Marker number (ignored for reference markers)

Parameters:
<TargetSearchVal> Target search value of marker no. `<Mk>`.
The value range and reset value depend on the selected target format (see `CALCulate<Chn>:MARKer<Mk>:SEARch:FORMat` or `CALCulate<Chn>:MARKer<Mk>:REFerence:SEARch:FORMat`).

Example:
```
CALC:MARK ON
```
Create marker no. 1 and display it in the center of the sweep range.
```
:CALC:MARK:TARG -10; FUNC:EXEC TARG
```
Define a target search value of -10 dB and start the target search.
```
CALC:MARK:X?
```
Query the stimulus value corresponding to the target search result.

Manual operation: See "Target Value" on page 516

CALCulate<Chn>:MARKer<Mk>:THReshold <Value>
CALCulate<Chn>:MARKer<Mk>:REFerence:THReshold <TargetSearchVal>

Defines a marker-specific threshold value for (single) peak searches with the related marker (see "Marker addressing" on page 1032) .

Use `CALCulate<Chn>:MARKer<Mk>:THReshold:STATe` or `CALCulate<Chn>:MARKer<Mk>:REFerence:THReshold:STATe` to activate it.

Suffix:
<Chn> Channel number used to identify the active trace

<Mk> Marker number (ignored for reference markers)

Parameters:
<Value> Threshold value
The unit is derived from the active trace format and cannot be modified. Changing the trace format resets the threshold to a format-specific default value.

Manual operation: See "Threshold Settings" on page 513

CALCulate<Chn>:MARKer<Mk>:THReshold:STATe <Active>
CALCulate<Chn>:MARKer<Mk>:REFerence:THReshold:STATe <Boolean>

Activates or deactivates the marker-specific threshold for (single) peak searches with the related marker (see "Marker addressing" on page 1032) .

Use `CALCulate<Chn>:MARKer<Mk>:THReshold` or `CALCulate<Chn>:MARKer<Mk>:REFerence:THReshold` to set the threshold value.

Suffix:
<Chn> Channel number used to identify the active trace

<Mk> Marker number (ignored for reference markers)

Parameters:
<Active> Boolean

Manual operation: See "Threshold Settings" on page 513

CALCulate<Chn>:MARKer<Mk>:TYPE <Mode>
CALCulate<Chn>:MARKer<Mk>:REFerence:TYPE <Mode>

Sets the marker mode for the related marker (see "Marker addressing" on page 1032). The marker must be created before using `CALCulate<Chn>:MARKer<Mk>[:STATe] ON` or `CALCulate<Chn>:MARKer<Mk>:REFerence[:STATe] ON`.

Suffix:

<Chn> Channel number used to identify the active trace

<Mk> Marker number (ignored for reference markers)

Parameters:

<Mode> NORMal | FIXed | ARBitrary

NORMal: If tracking is enabled (`CALCulate<Chn>:MARKer<Mk>:SEARch:TRACking ON` or `CALCulate<Chn>:MARKer<Mk>:REFerence:SEARch:TRACking ON`), the marker's stimulus value (`CALCulate<Chn>:MARKer<Mk>:X?` or `CALCulate<Chn>:MARKer<Mk>:REFerence:X?`) is updated automatically with every sweep, otherwise it is fixed. The marker position (`CALCulate<Chn>:MARKer<Mk>:Y?` or `CALCulate<Chn>:MARKer<Mk>:REFerence:Y?`) is adjusted to the corresponding response value, i.e. the marker is always positioned on the trace.
The marker's stimulus value can be set using `CALCulate<Chn>:MARKer<Mk>:X` or `CALCulate<Chn>:MARKer<Mk>:REFerence:X`; the marker automatically follows the trace.
FIXed: freezes the marker at the position determined by the current stimulus and response value. The response value stored with the marker is not adjusted to subsequent sweeps. Tracking is disabled. Stimulus and response value are stored with the marker; they are not adjusted to subsequent sweeps and trace format changes.
The marker stimulus can be set using `CALCulate<Chn>:MARKer<Mk>:X` or `CALCulate<Chn>:MARKer<Mk>:REFerence:X`, but the response value remains fixed.
ARBitrary: freezes the marker at the position determined by the current stimulus and response value. Tracking is disabled. The marker stores the stimulus value and – in addition – its X and Y coordinates in the current marker format (see `CALCulate<Chn>:MARKer<Mk>:FORMat` or `CALCulate<Chn>:MARKer<Mk>:REFerence:FORMat`).

The marker position can be set using CALCulate<Chn>:MARKer<Mk>:X and CALCulate<Chn>:MARKer<Mk>:Y or CALCulate<Chn>:MARKer<Mk>:REFerence:X and CALCulate<Chn>:MARKer<Mk>:REFerence:Y. If, in the current trace format, the X axis represents the stimulus, the marker's stimulus value is adjusted accordingly. Otherwise the marker's stimulus value remains unchanged. Switching between trace formats resets the marker position to the response value at the marker's stimulus value.

*RST: NORMal

Example:

```
CALC:MARK ON; :CALC:MARK:TYPE FIX
```

Create marker 1 and display it in the center of the sweep range as a fixed marker.

```
CALC:MARK:X 1GHz
```

Shift the marker horizontally. The response value remains fixed.

Manual operation: See "Marker Mode" on page 507

CALCulate<Chn>:MARKer<Mk>:X <StimulusValue>[, <Seg>[, <MeasPoint>]]
CALCulate<Chn>:MARKer<Mk>:REFerence:X <StimulusValue>[, <Seg>[, <MeasPoint>]]

Defines the stimulus value of the related marker (see "Marker addressing" on page 1032).

If the marker's mode is NORMal or FIXed (see CALCulate<Chn>:MARKer<Mk>:TYPE or CALCulate<Chn>:MARKer<Mk>:REFerence:TYPE), this command sets or gets the marker's stimulus value.

In ARBitrary mode, this is only true if the X axis represents the stimulus. For all other trace formats (see CALCulate<Chn>:FORMat) it sets or gets the X position of the marker, which is decoupled from the marker stimulus in this case.

Suffix:

<Chn> Channel number used to identify the active trace

<Mk> Marker number (ignored for reference markers)

Parameters:

<MarkerValue> If the marker mode of the related marker is ARBitrary and the trace format is complex (Polar, Smith, inverted Smith), this is the real part $Re(z_M)$ of the marker value z_M. In any other case, it is the marker's stimulus value.
If the marker mode is ARBitrary and the trace format is complex, the value range is -1 to +1. Otherwise -9.9E+11 Hz to +9.9E+11 Hz for frequency sweeps, -999 dBm to +999 dBm for power sweeps, 0 s to 127500 s for time sweeps and 1 to 100001 for CW mode.

Setting parameters:

<Seg> For a segmented frequency sweep with overlapping segments, you can assign the marker to a particular segment.
If specified, the <Seg> number must be valid and the <StimulusValue> must be inside the segment.

<Point> For a segmented frequency sweep with overlapping segments or with segments that have multiple sweep points at the same frequency, you can assign the marker to a particular segment <Seg> and sweep point <Point>.
If specified, the <Seg> and <Point> numbers must be valid and the <StimulusValue> must be inside the segment.

Example: Suppose that the active recall set contains an active trace no. 1 and the sweep range for a frequency sweep starts at 1 GHz.
`CALC:MARK ON`
Create marker no. 1 and display it in the center of the sweep range.
`CALC:MARK:X 1GHz`
Set the marker to the beginning of the sweep range.

Manual operation: See "Mkr <i> Stimulus / Ref Mkr Stimulus" on page 504

CALCulate<Chn>:MARKer<Mk>:Y <ResponseValue>
CALCulate<Chn>:MARKer<Mk>:REFerence:Y <RefResponseValue>

Sets or gets the (response) value of the related marker (see "Marker addressing" on page 1032).

The marker must be created before using `CALCulate<Chn>:MARKer<Mk>[:STATe] ON` or `CALCulate<Chn>:MARKer<Mk>:REFerence[:STATe] ON`.

Setting this value is only possible in ARBitrary mode (see `CALCulate<Chn>:MARKer<Mk>:TYPE` or `CALCulate<Chn>:MARKer<Mk>:REFerence:TYPE`. For NORMal and FIXed mode markers it is read-only.

Suffix:

<Chn> Channel number used to identify the active trace

<Mk> Marker number (ignored for reference markers)

Parameters:

<RefResponseValue> **Setting (ARBitrary mode only):** sets the vertical position of the marker in Y units of the current trace format.
Query: returns the marker position formatted according to the current marker format (see `CALCulate<Chn>:MARKer<Mk>:FORMat` or `CALCulate<Chn>:MARKer<Mk>:REFerence:FORMat`), i.e. as displayed in the marker info field (1 return value per row in the response column). Indetermined result values are returned as a sequence of dashes (------). This may occur in ARBitrary mode, if the transformation between trace format and marker format requires a concrete stimulus value.

Default unit: NN

Example: Suppose that the active recall set contains an active trace no. 1.
`CALC:MARK ON`
Create marker no. 1 and display it in the center of the sweep range.
`CALC:MARK:Y?`
Query the measurement value at the marker position.
`CALC:MARK:STAT:AREA LEFT, TOP`
For a subsequent check at the GUI or a hardcopy, move the info field to the top left position.

Manual operation: See "Mkr <i> Arb. Response / Ref Mkr Arb. Response" on page 504

7.3.1.9 CALCulate:MATH...

The `CALCulate:MATH...` commands permit processing of measured data in numerical expression format. The operators are +, -, *, / and use of constants and data arrays are permitted.

CALCulate<Chn>:MATH[:EXPRession]:SDEFine 1059
CALCulate<Chn>:MATH:FORMatted[:EXPRession]:SDEFine 1059
CALCulate<Chn>:MATH:FUNCtion 1060
CALCulate<Chn>:MATH:FORMatted:FUNCtion 1060
CALCulate<Chn>:MATH:MEMorize 1061
CALCulate<Chn>:MATH:STATe 1062
CALCulate<Chn>:MATH:FORMatted:STATe 1062
CALCulate<Chn>:MATH:WUNit[:STATe] 1063

CALCulate<Chn>:MATH[:EXPRession]:SDEFine <Expression>
CALCulate<Chn>:MATH:FORMatted[:EXPRession]:SDEFine <Expression>

Defines a mathematical trace for the active trace, using a string expression.

- The expression in the first command refers to raw, unformatted trace data (complex data). In order to apply it, `CALCulate<Chn>:MATH:FUNCtion` must be set to `NORMal` and `CALCulate<Chn>:MATH:STATe` must be set to `ON`.
- The expression in the second command refers to formatted trace data. In order to apply it, `CALCulate<Chn>:MATH:FORMatted:FUNCtion` must be set to `NORMal` and `CALCulate<Chn>:MATH:FORMatted:STATe` must be set to `ON`.

Trace math for unformatted and formatted trace data can be used at the same time.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<Expression> Operands, operators and functions; see table below.

Example: `*RST; :CALC:MATH:MEM`
Copy the current state of the default trace `'Trc1'` to a memory trace named `'Mem2[Trc1]'`. The memory trace is not displayed.
`CALC:MATH:SDEF 'Trc1 / Mem2[Trc1]'`
Define a mathematical trace, dividing the raw complex data trace by the stored memory trace. The mathematical trace is not displayed.
`CALC:MATH:STAT ON`
Display the mathematical trace instead of the active data trace.

Manual operation: See "Expression builder" on page 462

Expressions defined via `CALCulate<Ch>:MATH[:EXPRession]:SDEFine` or `CALCulate<Chn>:MATH:FORMatted[:EXPRession]:SDEFine` may contain the following elements:

Type	Complete List	Description
Operands	<Trace name> | activeTrc | Mem[activeTrc]	All traces and memory traces of the active recall set | Active trace | Active memory trace assigned to the active trace
Constants	e, pi | 1, -1.2, 8e9 | 1 + 2j, 2 + 1e-9j	Constants | Real values in decimal or exponential format | Complex numbers
Operators	- + , - , * , / , ^	Basic arithmetic operations; ^ for exponentiation
Functions	linMag (), dBMag (), Arg (), Re (), Im (), log (), ln (), tan (), atan (), sin (), asin (), cos (), acos (), Min (... , ...), Max (... , ...)	Mathematical functions with one or two arguments
Special Functions	StimVal	Current stimulus value (see description of operators for User Defined Math)
Brackets	( )	Priority of operations in complex expressions

CALCulate<Chn>:MATH:FUNCtion <Mode>
CALCulate<Chn>:MATH:FORMatted:FUNCtion <Mode>

Defines a simple mathematical trace based on the active trace and its active memory trace. The first command applies to raw, unformatted trace data (complex data), the second to formatted trace data. Both methods can be combined.

To apply the trace math, the corresponding mathematical mode must be switched on using `CALCulate<Chn>:MATH:STATe ON` and/or `CALCulate<Chn>:MATH:FORMatted:STATe ON`.

Note: Use `CALCulate<Chn>:MATH[:EXPRession]:SDEFine` and/or `CALCulate<Chn>:MATH:FORMatted[:EXPRession]:SDEFine` to define general math traces.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<Mode> NORMal | ADD | SUBTract | MULTiply | DIVide

NORMal
Math. trace = active data trace
When set to `NORMal`, the corresponding mathematical mode is turned `OFF`, otherwise it is turned `ON`.

ADD
Math. trace = data + memory

SUBTract
Math. trace = data - memory

MULTiply
Math. trace = data * memory

DIVide
Math. trace = data / memory

*RST: NORMal

Example: `*RST; :CALC:MATH:MEM`
Copy the current state of the default trace `'Trc1'` to a memory trace named `'Mem2[Trc1]'`. The memory trace is not displayed.
`CALC:MATH:FUNC DIV`
Define a mathematical trace, dividing the raw complex data trace trace by the stored memory trace. The mathematical trace is displayed instead of the active data trace.
`CALC:MATH:STAT?`
The response is 1 (mathematical mode for unformatted trace data switched on, mathematical trace displayed).

Manual operation: See "Data / <Mem>, Data - <Mem>" on page 461

CALCulate<Chn>:MATH:MEMorize

Copies the current state of the active data trace to a memory trace. If a mathematical trace is active, the data trace associated with the mathematical trace is copied. The memory trace is named `Mem<n>[<Data_Trace>]` where `<n>` counts all data and memory traces in the active recall set in chronological order, and `<Data_Trace>` is the name of the associated (copied) data trace.

The exact function of the command depends on the number of memory traces associated to the active data trace:

- If no memory trace is associated to the active trace, a new memory trace is generated.
- If several memory traces are associated to the active trace, the current measurement data overwrites the last generated or changed memory trace.

Note: To copy a trace to the memory without overwriting an existing memory trace or define a memory trace name, use `TRACe:COPY <MemTraceName>,<DataTraceName>`. To copy an active mathematical trace use `TRACe:COPY:MATH <MemTraceName>,<DataTraceName>`

Suffix:
<Chn> Channel number used to identify the active trace

Example:
```
*RST; :CALC:MATH:MEM
```
Copy the current state of the default trace `'Trc1'` to a memory trace named `'Mem2[Trc1]'`. The memory trace is not displayed.
```
DISP:WIND:TRAC2:FEED 'Mem2[Trc1]'
```
Display the created memory trace in the active diagram area (diagram area no. 1).

Usage: Event

Manual operation: See "Data to <Destination>" on page 456

CALCulate<Chn>:MATH:STATe <Boolean>
CALCulate<Chn>:MATH:FORMatted:STATe <Boolean>

Activates or deactivates trace math for unformatted and/or formatted trace data, i.e. the trace formulae defined via `CALCulate<Chn>:MATH[:EXPRession]:SDEFine` and/or `CALCulate<Chn>:MATH:FORMatted[:EXPRession]:SDEFine`, respectively.

When trace math is active either for unformatted or formatted data, the resulting mathematical trace is calculated and displayed instead of the active data trace.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<Boolean> ON – apply trace math on the
OFF – display the active data trace
*RST: OFF

Example:
```
*RST; :CALC:MATH:MEM
```
Copy the current state of the default trace `'Trc1'` to a memory trace named `'Mem2[Trc1]'`. The memory trace is not displayed.
```
CALC:MATH:SDEF 'Trc1 / Mem2[Trc1]'
```
Define a mathematical trace, dividing the complex data trace by the stored complex memory trace. The mathematical trace is not displayed.
```
CALC:MATH:STAT ON
```
Display the mathematical trace instead of the active data trace.

Manual operation: See "Trace Math" on page 460

CALCulate<Chn>:MATH:WUNit[:STATe] <Boolean>

Controls the conversion and formatting of the mathematic expression defined via `CALCulate<Chn>:MATH[:EXPRession]:SDEFine`.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<Boolean> ON –"Result is Wave Quantity" enabled: the analyzer assumes that the result of the mathematical expression represents a voltage.
OFF –"Result is Wave Quantity" disabled: the analyzer assumes that the result of the mathematical expression is dimensionless.
*RST: OFF

Example:
```
*RST; SWE:TYPE POW
CALC:PAR:SDEF 'Trc1', 'a1'
```
Reset the instrument, activate a power sweep, and select a wave quantity a_1 for the trace `Trc1`.
```
DISP:WIND:TRAC:FEED 'Trc1'
```
Display the generated trace in the active window.
```
CALC:MATH:SDEF 'StimVal'; STAT ON
```
Define a mathematical trace and display it instead of the active data trace.
```
CALC:MATH:WUN ON
```
Take into account that the stimulus value is a voltage (derived from the source power) rather than a dimensionless quantity. The y-axis range of the mathematical trace now exactly corresponds to the power sweep range.

Manual operation: See "Result is Wave Quantity" on page 464

7.3.1.10 CALCulate:PARameter...

The `CALCulate:PARameter...` commands assign names and measurement parameters to traces. The commands are device-specific.

CALCulate<Ch>:PARameter:CATalog? 1064
CALCulate<Ch>:PARameter:CATalog:SENDed? 1064
CALCulate<Ch>:PARameter:COPY 1064
CALCulate<Ch>:PARameter:COPY:CHANnel 1065
CALCulate<Ch>:PARameter:DEFine:SGRoup 1065
CALCulate<Ch>:PARameter:DELete 1067
CALCulate:PARameter:DELete:ALL 1068
CALCulate<Ch>:PARameter:DELete:CALL 1068
CALCulate<Ch>:PARameter:DELete:CMEMory 1068
CALCulate:PARameter:DELete:MEMory 1068
CALCulate<Ch>:PARameter:DELete:SGRoup 1068
CALCulate<Ch>:PARameter:MEASure 1069
CALCulate<Ch>:PARameter:MEASure:SENDed 1069

CALCulate:PARameter:MOVE....1070
CALCulate:PARameter:NEW:CHANnel....1070
CALCulate<Ch>:PARameter:SDEFine....1071
CALCulate<Ch>:PARameter:SDEFine:SENDed....1074
CALCulate<Ch>:PARameter:SELect....1074

CALCulate<Ch>:PARameter:CATalog?

Returns the trace names and measurement parameters of all traces assigned to a particular channel.

The result is a string containing a comma-separated list of trace names and measurement parameters, e.g. `'CH4TR1,S11,CH4TR2,S12'`. The measurement parameters are returned according to the naming convention of `CALCulate<Ch>:PARameter:SDEFine`. The order of traces in the list reflects their creation time: The oldest trace is the first, the newest trace is the last trace in the list.

Suffix:
<Ch> Channel number. If unspecified the numeric suffix is set to 1.

Example: `CALC4:PAR:SDEF 'Ch4Tr1', 'S11'`
Create channel 4 and a trace named Ch4Tr1 to measure the input reflection coefficient S_{11}.
`CALC4:PAR:CAT?`
Query the traces assigned to channel 4. If Ch4Tr1 is the only trace assigned to channel 4, the response is `'CH4TR1,S11'`.

Usage: Query only

CALCulate<Ch>:PARameter:CATalog:SENDed?

Returns the trace names and measurement parameters of all traces assigned to a particular channel.

Similar to `CALCulate<Ch>:PARameter:CATalog?`, but in presence of balanced ports this command distinguishes between

- single-ended S-parameters for **logical** ports (`SSS<Li><Lj>`)
- "raw" single-ended S-parameters referring to **physical** ports (`S<Pi><Pj>`)

Suffix:
<Ch> Channel number. If unspecified the numeric suffix is set to 1.

Usage: Query only

CALCulate<Ch>:PARameter:COPY <TraceName>[, <ToNewDiag>]

Copies the active trace of channel <Chn>:

- To channel <Chn>
- With name <TraceName>
- To the same or to a new diagram

Suffix:
<Ch> Channel number used to identify the active trace

Parameters:
<TraceName> Name of the new trace (i.e. the copy).
Note that trace names must be unique within the active setup.

<ToNewDiag> Boolean

OFF (0)
The new trace is assigned to the diagram area of the copied trace (default, if omitted).
Corresponds to the "Add Trace" GUI function.

ON (1)
The new trace is assigned to a new diagram area.
Corresponds to the "Add Trace + Diagram" GUI function.

Manual operation: See "Add Trace" on page 451

CALCulate<Ch>:PARameter:COPY:CHANnel [<ToNewDiag>]

Copies channel <Ch> with all its settings (including a possible channel calibration) and traces, and optionally creates a diagram for the new channel.

Suffix:
<Ch> Channel number

Parameters:
<ToNewDiag> **OFF | 0**
The created traces are displayed in the active diagram area (default).

ON | 1
The created traces are displayed in a new diagram area.

Manual operation: See "Copy Channel" on page 671

CALCulate<Ch>:PARameter:DEFine:SGRoup <LogicalPort1>[, <LogicalPort2>]...

Creates the traces for all S-parameters associated with a group of logical ports (S-parameter group). The traces can be queried using `CALCulate<Ch>:DATA:SGRoup?`.

Traces must be selected to become active traces; see `CALCulate<Ch>:PARameter:SELect`.

Note: Each channel can contain a single S-parameter group only. Defining a new S-parameter group deletes the previous one. Use `CALCulate<Ch>:PARameter:DELete:SGRoup` on page 1068 to delete the current S-group explicitly.

Suffix:

<Ch> Channel number. <Ch> may be used to reference a previously defined channel. If <Ch> does not exist, it is generated with default channel settings.

Parameters:

<LogicalPort1> Logical (balanced or unbalanced) port numbers. The port numbers must be in ascending order, their number is limited by the test ports of the analyzer. With n logical port numbers, the command generates n^2 traces. The traces correspond to the following S-parameters:
$S_{<log_port1><log_port1>}$, $S_{<log_port1><log_port2>}$ ... $S_{<log_port1><log_port<n>>}$
...
$S_{<log_port<n>><log_port1>}$, $S_{<log_port<n>><log_port2>}$... $S_{<log_port<n>><log_port<n>>}$,
e.g. S_{11}, S_{12}, S_{21}, S_{22} for <log_port1> = 1, <log_port2> = 2. If only one logical port <log_port1> is specified, a single trace with the reflection coefficient $S_{<log_port1><log_port1>}$ is created.

Trace names

The generated traces are assigned the following trace names:
<Ch_name>_SG_S<log_port1><log_port1>,
<Ch_name>_SG_S<log_port1><log_port2> ...
<Ch_name>_SG_S<log_port1><log_port<n>> ...<Ch_name>_SG_S<log_port<n>><log_port1>,
<Ch_name>_SG_S<log_port<n>><log_port2>...
<Ch_name>_SG_S<log_port<n>><log_port<n>>,
e.g. Ch1_SG_S11, Ch1_SG_S12, Ch1_SG_S21, Ch1_SG_S22 for <Ch_name> = Ch1, <log_port1> = 1, <log_port2> = 2. The trace names are displayed in the "Channel Manager" and in the "Trace Manager" dialogs where they can be changed manually. The <Ch_name> is defined via
`CONFigure:CHANnel<Ch>:NAME '<Ch_name>'`.
Trace names are important for referencing the generated traces; see program example below.

<LogicalPort2>

Example: `CALC2:PAR:DEF:SGR 1,2`
Create channel 2 and four traces to measure the two-port S-parameters S_{11}, S_{12}, S_{21}, S_{22}. The traces are not displayed.
```
DISP:WIND:TRAC2:FEED 'Ch2_SG_S11'
DISP:WIND:TRAC3:FEED 'Ch2_SG_S12'
DISP:WIND:TRAC4:FEED 'Ch2_SG_S21'
DISP:WIND:TRAC5:FEED 'Ch2_SG_S22'
```
Display the four traces in the diagram no. 1.
```
INIT2:CONT OFF; :INIT2:IMMediate; *OPC
```
Perform a complete speep in channel no. 2 to ensure the traces are completely "filled" with data.
```
CALC2:DATA:SGR? SDAT
```
Retrieve all four traces as unformatted data (real and imaginary part at each sweep point). The analyzer first returns the complete S_{11} trace, followed by the S_{12}, S_{21}, and S_{22} traces.
```
CALC2:PAR:DEL:SGR
```
Delete the previously created port group.

Manual operation: See "All S-Params" on page 294

CALCulate<Ch>:PARameter:DELete <TraceName>

Deletes a trace with a specified trace name and channel.

Suffix:
<Ch> Channel number.

Setting parameters:
<TraceName> Trace name, e.g. 'Trc4'. See "Rules for trace names" in Section 5.5.1.3, "Trace Manager dialog", on page 453.

Example: `CALCulate4:PARameter:SDEFine 'Ch4Tr1', 'S11'`
Create channel 4 and a trace named `Ch4Tr1` to measure the input reflection coefficient S_{11}.
```
CALCulate4:PAR:CAT?
```
Query the traces assigned to channel 4. `Ch4Tr1` is the only trace assigned to channel 4, so the response is `'CH4TR1,S11'`.
```
CALCulate4:PARameter:SDEFine 'CH4TR2', 'S21';
SDEFine 'CH4TR3', 'S12'; SDEFine 'CH4TR4',
'S22'
```
Create three more traces for the remaining 2-port S-parameters.
```
CALCulate4:PARameter:DELete 'CH4TR1'
```
Delete the first created trace.
```
CALCulate4:PARameter:DELete:CALL
```
Delete the remaining three traces in channel 4.
```
CALCulate:PARameter:DELete:ALL
```
Delete all traces, including the default trace `Trc1` in channel 1.

Usage: Setting only

Manual operation: See "Delete Trace" on page 451

CALCulate:PARameter:DELete:ALL

Deletes all traces in all channels of the active recall set, including the default trace `Trc1` in channel 1. The manual control screen shows "No Trace".

Example: See `CALCulate<Ch>:PARameter:DELete`

Usage: Event

Manual operation: See "Delete Trace" on page 451

CALCulate<Ch>:PARameter:DELete:CALL

Deletes all traces in channel no. `<Ch>`.

Suffix:
<Ch> Channel number

Example: See `CALCulate<Ch>:PARameter:DELete`

Usage: Event

Manual operation: See "Delete Trace" on page 451

CALCulate<Ch>:PARameter:DELete:CMEMory

Deletes all memory traces in channel <Ch>.

Suffix:
<Ch> Channel number

Usage: Event

CALCulate:PARameter:DELete:MEMory

Deletes all memory traces in all channels.

Usage: Event

Manual operation: See "Delete All Mem" on page 459

CALCulate<Ch>:PARameter:DELete:SGRoup

Deletes a group of logical ports (S-parameter group), previously defined via `CALCulate<Ch>:PARameter:DEFine:SGRoup`.

Suffix:
<Ch> Channel number. `<Ch>` may be used to reference a previously defined channel. If `<Ch>` does not exist, it is generated with default channel settings.

Example: See `CALCulate<Ch>:PARameter:DEFine:SGRoup`

Usage: Event

CALCulate<Ch>:PARameter:MEASure <TraceName>, <Result>

Assigns a measurement result to an **existing** trace. The query returns the result assigned to the specified trace (no second parameter; see example).

Note: To create a new trace and at the same time assign the attributes, use `CALCulate<Ch>:PARameter:SDEFine`. To display the trace, create a diagram (`DISPlay[:WINDow<Wnd>][:STATe] ON`) and assign the trace to this diagram (`DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:FEED`); see example below.

Traces must be selected to become active traces; see `CALCulate<Ch>:PARameter:SELect`. `CALCulate<Ch>:PARameter:CATalog?` returns a list of all defined traces. You can open the "Trace Manager" dialog to obtain an overview of all channels and traces, including the traces that are not displayed.

Suffix:

<Ch> Channel number of an existing channel containing the referenced trace.

Parameters:

<TraceName> Trace name, string variable, e.g. 'Trc4'. See "Rules for trace names" in "Table Area" on page 453. Trace names must be unique across all channels and diagrams.

<Result> Measurement parameter (string variable); see Table 7-3.
A query of a wave quantity `'xy'` returns `'xyD<n><Detector>'`, where `<n>` numbers the source (drive) port, and `<Detector>` denotes the detector setting. A query of a ratio `'x/y'` returns `'xD<n>/yD<m><Detector>'`, where `<n>` and `<m>` number the source ports

Example:

```
CALC4:PAR:SDEF 'Ch4Tr1', 'S11'
```
Create channel 4 and a trace named `Ch4Tr1` to measure the input reflection coefficient S_{11}.

```
CALC4:PAR:MEAS 'Ch4Tr1', 'a1'
```
Change the measurement parameter of the trace and measure the wave quantity a_1.

```
CALC4:PAR:MEAS? 'Ch4Tr1'
```
Query the measured quantity. The response is `'A1D1SAM'`.

Manual operation: See "S-Parameter (selector)" on page 293

CALCulate<Ch>:PARameter:MEASure:SENDed <TraceName>, <Result>

Assigns a measurement result to an **existing** trace. The query returns the result assigned to the specified trace (no second parameter; see example).

Similar to `CALCulate<Ch>:PARameter:MEASure`, but in presence of balanced ports this command distinguishes between

- single-ended S-parameters for **logical** ports (`SSS<Li><Lj>`)
- "raw" single-ended S-parameters referring to **physical** ports (`S<Pi><Pj>`)

Suffix:
<Ch> Channel number

Setting parameters:
<TraceName> Trace name

<Result> Measured quantity

Manual operation: See "S-Parameter (selector)" on page 293

CALCulate:PARameter:MOVE <TraceName>, <ChannelName>

Moves trace <TraceName> to channel <ChannelName>.

Setting parameters:
<TraceName> Trace name
Use `CONFigure:TRACe:CATalog?` to query for existing traces.

<ChannelName> Channel name
Use `CONFigure:CHANnel:CATalog?` to query for existing channels.

Usage: Setting only

Manual operation: See "Channel" on page 450

CALCulate:PARameter:NEW:CHANnel [<ToNewDiag>]

Creates a channel and trace with default settings, and optionally creates a diagram for the new channel.

Use:

- `CONFigure:CHANnel<Ch>[:STATe] ON` to create a channel based on the settings of the previously **active channel** (if any)
- `CALCulate<Ch>:PARameter:COPY:CHANnel [OFF]` to copy a channel including its traces

Firmware version 4.0 and higher.

Setting parameters:
<ToNewDiag> **OFF | 0**
The created traces are displayed in the active diagram area (default).

ON | 1
The created traces are displayed in a new diagram area.

Usage: Setting only

Manual operation: See "New Channel" on page 671

CALCulate<Ch>:PARameter:SDEFine <TraceName>, <Result>

Creates a trace and assigns a channel number, a name and a measurement parameter to it. The trace becomes the active trace in the channel but is not displayed.

Note: To display the trace defined via `CALCulate<Ch>:PARameter:SDEFine`, create a diagram (`DISPlay[:WINDow<Wnd>][:STATe] ON`) and assign the trace to this diagram (`DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:FEED`); see example below. `CALCulate<Ch>:PARameter:MEASure` changes the measurement result of an existing trace.

To select an existing trace as the active trace, use `CALCulate<Ch>:PARameter:SELect`. You can open the trace manager to obtain an overview of all channels and traces, including the traces that are not displayed.

Tip: This command has no query form. Use `CALCulate<Ch>:PARameter:MEASure <TraceName>` to query the measurement result of the trace. `CALCulate<Ch>:PARameter:CATalog?` returns a list of all defined traces.

Suffix:

<Ch> Channel number
If channel <Ch> does not exist, it is created based on the settings of the **active channel** (if any). If you need a channel with default settings, use `CALCulate:PARameter:NEW:CHANnel` first.

Setting parameters:

<TraceName> Trace name, string variable, e.g. 'Trc4'. See "Rules for trace names" in "Table Area" on page 453. Trace names must be unique across all channels and diagrams.
If a trace with the selected trace name already exists, the analyzer behaves as follows: If the existing trace is assigned to the same channel as the new trace, it is deleted. The new trace is not automatically assigned to a diagram area; see note above. If the existing trace is assigned to a different channel, no new trace can be created. The analyzer returns an error message.

<Result> Measurement result (string variable), see table below.

Example:

```
CALC4:PAR:SDEF 'Ch4Tr1', 'S11'
```

Create channel 4 and a trace named Ch4Tr1 to measure the input reflection coefficient S_{11}.

```
DISP:WIND2:STAT ON
```

Create diagram area no. 2.

```
DISP:WIND2:TRAC:FEED 'CH4TR1'
```

Display the generated trace in diagram area no. 2.

Usage: Setting only

Manual operation: See "S-Parameter (selector)" on page 293

Table 7-3: String identifiers for measurement results

Note: All port numbers in a result identifier refer to **logical (=DUT) ports**; to avoid ambiguities they must be represented by the same number of digits (e.g. S21 or S0201). The valid port numbers are determined by the channel's logical port configuration For details about the measurement results, see Section 4.3, "Measurement results", on page 116.	
'S11' | 'S12' | ... | 'S0101' | ...	Single-ended S-parameters S<i><j> for logical (DUT) ports <i> and <j>. To avoid ambiguities, <i> and <j> must be specified with the same number of digits.
'SCD11' | ...	S-parameters involving balanced ports must be specified in the form S<m_i><m_j><i><j>, where <m_i> and <m_j> denote the port modes of the related logical ports <i> and <j>. In general, for the port modes <m_i><m_j> all pairs of D (differential, balanced), C (common, balanced) and S (single-ended, unbalanced) are allowed.
'Y11' | ... | 'YSS11' | ... | 'YCC11' | ... | 'YDD11' | 'Z11' | ... | 'ZSS11' | ... | 'ZCC11' | ... | 'ZDD11' | ...	Short-circuit Y-parameters and open-circuit Z-parameters with port modes and port numbers like for normal mode S-parameters. Selecting a parameter Y...<n><m> or Z...<n><m> sets the range of logical port numbers to be considered for the Y and Z-parameter measurement to <n>:<m>.
'Y-S11' | ... | 'Y-SSS11' | ... | 'Y-SCC11' | ... | 'Y-SDD11' | 'Z-S11' | ... | 'Z-SSS11' | ... | 'Z-SCC11' | ... | 'Z-SDD11' | ...	S-parameters converted to matched-circuit admittances and impedances with port modes and port numbers like for normal mode S-parameters. For transmission parameters, this refers to series resistances.
'Y-S12SER' | 'Y-S12PAR' | ... | 'Z-S12SER' | 'Z-S12PAR' | ... | 'C-S12SER' | 'C-S12PAR' | ... | 'L-S12SER' | 'L-S12PAR' | ... | 'R-S12SER' | 'R-S12PAR' | ...	Converted transmission admittances, impedances, capacitances, inductances and resistances, calculated as seriesSection 4, "Concepts and features", on page 79 or parallel admittances/impedances.
'C-S12SER' | 'C-S12PAR' | ... | 'L-S12SER' | 'L-S12PAR' | ... | 'R-S12SER' | 'R-S12PAR' | ...	Converted transmission capacitances, inductances and resistances, calculated from the corresponding converted impedances.
'SHUNT-S12' | 'SHUNT-S13 | ... | 'SHUNT-S21' | 'SHUNT-S23' | ...	Converted impedances, measured using the shunt-thru method.
'C-S12SHUNT' | 'L-S12SHUNT' | 'R-S12SHUNT' | ...	Converted transmission capacitances, inductances and resistances, calculated from the corresponding converted impedances, measured using the shunt-thru method.
'A1' | ... | 'A01' | ... | 'B1' | ... | 'B01' | ... ... | 'A1SAM' | 'A1AVG' | 'A1AMP'	Wave quantities A<meas>D1 (for a wave) and B<meas>D1. The strings SAM, AVG, AMP appended to the wave quantities denote a normal (sample, SAM), AVG Real Imag (AVG), or AVG Mag Phase (AMP) detector. The observation time for average detectors is set via `[SENSe<Ch>:]SWEep:DETector:TIME`.
'A1D2' | ... | 'A01D02' | ... | 'B1D2' | ... | 'B01D02' | ... ... | 'A1D1SAM' | 'A1D1AVG' | 'A1D1AMP' | ...	Wave quantities A<meas>D<drive> (for a wave) and B<meas>D<drive> (for b wave) for logical ports <meas> and <drive>. The strings SAM, AVG, AMP appended to the wave quantities denote a normal (sample, SAM), AVG Real Imag (AVG), or AVG Mag Phase (AMP) detector.
'A1G2' | ... | 'A01G02' | ... | 'B1G2' | ... | 'B01G02' | ... | 'A1G1SAM' | 'A1G1AVG' | 'A1G1AMP' | ...	Wave quantities with an external generator providing the stimulus signal (G<no> for generator no.). The strings SAM, AVG, AMP appended to the wave quantities denote a normal (sample, SAM), AVG Real Imag (AVG), or AVG Mag Phase (AMP) detector.

'B2/A1' | ... 'B02/A01' | ... ... | 'B2/A1SAM' | 'B2/A1AVG' | B2/A1AMP' | ...	Ratio of wave quantities with port numbers like for normal mode S-parameters. The strings SAM, AVG, AMP appended to the wave quantities denote a normal (sample, SAM), AVG Real Imag (AVG), or AVG Mag Phase (AMP) detector.
'B2D1/A1D1' | ... 'B02D01/A01D01' | ... ... | 'B2D1/A1D1SAM' | 'B2D1/A1D1AVG' 'B2D1/A1D1AMP' | ...	Ratios of wave quantities with drive ports The strings SAM, AVG, AMP appended to the wave quantities denote a normal (sample, SAM), AVG Real Imag (AVG), or AVG Mag Phase (AMP) detector.
'B2G1/A1G1' | ... 'B02G01/A01G01' | ... 'B2G1/A1G1SAM' | 'B2G1/A1G1AVG' 'B2G1/A1G1AMP' | ...	Ratios of wave quantities with port numbers and external generator providing the stimulus signal (G<no> for generator no.). The strings SAM, AVG, AMP appended to the wave quantities denote a normal (sample, SAM), AVG Real Imag (AVG), or AVG Mag Phase (AMP) detector.
'IMB21' | 'IMB12' | 'IMB31' | ...	Imbalance parameter Imb<receive_port><drive_port> for logical port numbers <receive_port> and <drive_port>. The logical ports must be different and at least one of them must be balanced.
'CMRR11' | 'CMRR21' | 'CMRR12' | 'CMRR22' | ...	Common mode rejection ratio CMRR<receive_port><drive_port> parameter for logical port numbers <receive_port> and <drive_port>, at least one of them balanced.
'IMB1-23' | 'IMB23-1' | 'IMB1-24' | ...	Differential Imbalance parameters between a balanced and two single-ended logical ports
'KFAC21' | 'KFAC12' | ...	Stability factor K (for unbalanced ports only)
'MUF121' | 'MUF112' | ...	Stability factor 1 (for unbalanced ports only)
'MUF221' | 'MUF212' | ...	Stability factor 2 (for unbalanced ports only)
'Pmtr1G1' | 'Pmtr2G1' | 'Pmtr3D1' | ...	Power sensor measurement using a power meter 'Pmtr<no>' and an external generator 'G<no>'
'Pmtr1D1' | 'Pmtr1D2', ..., 'Pmtr2D1' | 'Pmtr2D2', ...	Power sensor measurement using a power meter 'Pmtr<m>' and analyzer source port 'D<n>'
'DC1D1' ... 'DC4D1' | 'DC1D01' ... 'DC4D01' 'DC1V' | 'DC10V'	DC measurement DC 1 to DC 4. D1 or D01 etc. denotes the (logical) drive port. For R&S ZVAB compatibility, the query returns 'DC1D1' or 'DC2D1'.
'PAE21' | PAE12' | ...	Power added efficiency (referring to logical ports)
'IM3UO' | 'IM3LO' | 'IM3MO' | 'IM5UO' | ... | 'IM9MO'	Intermodulation product IM<order><side>O, where <order> = 3 | 5 | 7 | 9, <side> = U | L | M (for upper or lower or major), at DUT output
'IM3UOR' | 'IM3LOR' | 'IM3MOR' | 'IM5UOR' | ... | 'IM9MOR'	Intermodulation product (as explained above), displayed in dB units relative to the measured lower tone level at DUT output ("intermodulation suppression")
'IP3UO' | 'IP3LO' | 'IP3MO' | 'IP5UO' | ... | 'IP9MO'	Intercept point IP<order><side>O where <order> = 3 | 5 | 7 | 9, <side> = U | L | M (for upper or lower or major), at DUT output
UTI | UTO | LTI | LTO	Upper or lower tone at DUT input or output
NLO	Intermodulation measurements: Noise level at DUT output Requires options R&S ZNB-K14 and R&S ZNB-K30
'NF12' | 'NF13' | ... | 'NF21' | 'NF23' | ...	Noise figure parameters NF<out><in>, where <out> and <in> denote the output and input port numbers of the DUT, and <out>≠<in>. Requires option R&S ZNB-K30.

'b1/a2' | 'b1/a3' | ... | 'b2/a1' | 'b2/a3' | ...	Gain of the DUT, as measured in noise figure channels Requires option R&S ZNB-K30.
'NF1R' | 'NF2R' | ...	Receiver noise figure, as measured in noise figure channels Requires option R&S ZNB-K30.
Cu(P<Src Port>)<_lowercase_ext_dll_name>/<Task Type>/<Additional Input>	Custom traces, calculated using external DLL `<ext_dll_name>.dll`.

*) Selecting a parameter Y...<n><m> or Z...<n><m> sets the range of port numbers to be considered for the Y and Z-parameter measurement to <n>:<m>.

CALCulate<Ch>:PARameter:SDEFine:SENDed <TraceName>, <Result>

Creates a trace and assigns a channel number, a name and a measurement parameter to it. The trace becomes the active trace in the channel but is not displayed.

Similar to `CALCulate<Ch>:PARameter:SDEFine`, but in presence of balanced ports this command distinguishes between

- single-ended S-parameters for **logical** ports (`SSS<Li><Lj>`)
- "raw" single-ended S-parameters referring to **physical** ports (`S<Pi><Pj>`)

Suffix:

<Ch> Channel number
If channel <Ch> does not exist, it is created based on the settings of the **active channel** (if any). If you need a channel with default settings, use `CALCulate:PARameter:NEW:CHANnel` first.

Setting parameters:

<TraceName> Trace name

<Result> Measured quantity
See `CALCulate<Ch>:PARameter:SDEFine` on page 1071

Usage: Setting only

Manual operation: See "S-Parameter (selector)" on page 293

CALCulate<Ch>:PARameter:SELect <TraceName>

Selects an existing trace as the active trace of the channel. All trace commands without explicit reference to the trace name act on the active trace (e.g. `CALCulate<Chn>:FORMat`). `CALCulate<Ch>:PARameter:SELect` is also necessary if the active trace of a channel has been deleted.

Suffix:

<Ch> Channel number.

Parameters:

<TraceName> Trace name, e.g. 'Trc4'. See "Rules for trace names" in "Table Area" on page 453.

Example:
`CALC4:PAR:SDEF 'Ch4Tr1', 'S11'`
Create channel 4 and a trace named `Ch4Tr1` to measure the input reflection coefficient S_{11}. The trace is the active trace in channel 4.
`CALC4:PAR:SDEF 'Ch4Tr2', 'S22'`
Create another trace named `Ch4Tr2` to measure the output reflection coefficient S_{22}. Again this new trace becomes the active trace in channel 4.
`CALC4:PAR:SEL 'Ch4Tr1'`
Select the first trace `Ch4Tr1` as the active trace.
`CALC4:FORM MLIN`
Calculate the magnitude of S_{11} and display it in a linearly scaled Cartesian diagram.

Manual operation: See "Active Trace" on page 450

7.3.1.11 CALCulate:PULSe...

The commands are related to the "Pulse Response" functionality of the "Extended Time Domain" application.

CALCulate<Chn>:PULSe:DUT:MODE....1075
CALCulate<Chn>:PULSe:EMPHasis:PRE....1076
CALCulate<Chn>:PULSe:EMPHasis:PST1....1076
CALCulate<Chn>:PULSe:EMPHasis:PST2....1076
CALCulate<Chn>:PULSe:EMPHasis:STATe....1077
CALCulate<Chn>:PULSe:EQUalization:CTLE:DC....1077
CALCulate<Chn>:PULSe:EQUalization:CTLE:POL1....1077
CALCulate<Chn>:PULSe:EQUalization:CTLE:POL2....1077
CALCulate<Chn>:PULSe:EQUalization:CTLE:RATio....1078
CALCulate<Chn>:PULSe:EQUalization:CTLE:ZERO....1078
CALCulate<Chn>:PULSe:EQUalization:STATe....1078
CALCulate<Chn>:PULSe:INPut:AMPLitude....1078
CALCulate<Chn>:PULSe:INPut:DRATe....1079
CALCulate<Chn>:PULSe:INPut:PWIDth....1079
CALCulate<Chn>:PULSe:INPut:SLICes....1079
CALCulate<Chn>:PULSe:MEASurement:CURSors:POST....1079
CALCulate<Chn>:PULSe:MEASurement:CURSors:PRE....1080
CALCulate<Chn>:PULSe:MEASurement:MAXTime....1080
CALCulate<Chn>:PULSe:MEASurement:STATe....1080
CALCulate<Chn>:PULSe:MEASurement:TYPE....1081
CALCulate<Chn>:PULSe:STATe....1081

CALCulate<Chn>:PULSe:DUT:MODE <DUTMode>

Allows you to (temporarily) switch between the measured DUT and an ideal one (with flat frequency response) in the related pulse response diagram.

Suffix:
<Chn> The channel number used to identify the active trace.

Parameters:
<DUTMode> IDEal | MEASured
IDEal | MEASured

Options: R&S ZNB-K20

Manual operation: See "DUT" on page 408

CALCulate<Chn>:PULSe:EMPHasis:PRE <PRE>

Sets the weight of the pre-cursor tap for the pre-emphasis FIR filter in the calculation chain of the related pulse response diagram.

Suffix:
<Chn> The channel number used to identify the active trace.

Parameters:
<PRE> Weight relative to the "Cursor" tap.

Options: R&S ZNB-K20

Manual operation: See "Emphasis" on page 407

CALCulate<Chn>:PULSe:EMPHasis:PST1 <PST1>

Sets the weight of the first postcursor tap for the pre-emphasis FIR filter in the calculation chain of the related pulse response diagram.

Suffix:
<Chn> The channel number used to identify the active trace.

Parameters:
<PST1> Weight relative to the "Cursor" tap.

Options: R&S ZNB-K20

Manual operation: See "Emphasis" on page 407

CALCulate<Chn>:PULSe:EMPHasis:PST2 <PST2>

Sets the weight of the second postcursor tap for the pre-emphasis FIR filter in the calculation chain of the related pulse response diagram.

Suffix:
<Chn> The channel number used to identify the active trace.

Parameters:
<PST2>

Options: R&S ZNB-K20

Manual operation: See "Emphasis" on page 407

CALCulate<Chn>:PULSe:EMPHasis:STATe <Boolean>

Activates/deactivates pre-emphasis in the calculation chain of the related pulse response diagram.

Suffix:
<Chn> The channel number used to identify the active trace.

Parameters:
<Boolean> ON | OFF - enables or disables pre-emphasis for pulse response measurements.

Options: R&S ZNB-K20

Manual operation: See "Emphasis" on page 407

CALCulate<Chn>:PULSe:EQUalization:CTLE:DC <Gain>

Specifies the DC gain of the CTLE (a two-pole filter with single zero) at the receiver simulation of the related pulse response diagram.

Suffix:
<Chn> The channel number used to identify the active trace.

Parameters:
<Gain> *RST: 0 dB
Default unit: dB

Options: R&S ZNB-K20

Manual operation: See "Equalization" on page 408

CALCulate<Chn>:PULSe:EQUalization:CTLE:POL1 <POL1>

Defines the frequency position for Pole 1 of the equalization filter.

Suffix:
<Chn> The channel number used to identify the active trace.

Parameters:
<POL1> Frequency value of pole 1

Options: R&S ZNB-K20

Manual operation: See "Equalization" on page 408

CALCulate<Chn>:PULSe:EQUalization:CTLE:POL2 <POL2>

Defines the frequency position for Pole 2 of the equalization filter.

Suffix:
<Chn> The channel number used to identify the active trace.

Parameters:
<POL2> Frequency value of pole 2

Options: R&S ZNB-K20

Manual operation: See "Equalization" on page 408

CALCulate<Chn>:PULSe:EQUalization:CTLE:RATio <Ratio>

Ratio of the output to input voltage.

Suffix:
<Chn> The channel number used to identify the active trace.

Parameters:
<Ratio> Numeric value.

Options: R&S ZNB-K20

Manual operation: See "Equalization" on page 408

CALCulate<Chn>:PULSe:EQUalization:CTLE:ZERO <ZERO>

Defines the frequency position for the zero transition of the equalization filter.

Suffix:
<Chn> The channel number used to identify the active trace.

Parameters:
<ZERO> Frequency value of the zero position.

Options: R&S ZNB-K20

Manual operation: See "Equalization" on page 408

CALCulate<Chn>:PULSe:EQUalization:STATe <Boolean>

Enables or disables equalization for the pulse response measurements.

Suffix:
<Chn> The channel number used to identify the active trace.

Parameters:
<Boolean> ON | OFF - enables or disables equalization.

Options: R&S ZNB-K20

Manual operation: See "Equalization" on page 408

CALCulate<Chn>:PULSe:INPut:AMPLitude <AMPL>

Defines the maximum voltage level of the simulated digital signal used for the pulse response measurements.

Suffix:
<Chn> The channel number used to identify the active trace.

Parameters:
<AMPL> The pulse amplitude. Default unit: V

Options: R&S ZNB-K20

Manual operation: See "Amplitude" on page 407

CALCulate<Chn>:PULSe:INPut:DRATe <Data>

The number of generated pulses per second.

Suffix:
<Chn> The channel number used to identify the active trace.

Parameters:
<Data> Specifies the pulse rate with units BPS, KBPS (=10^3 BPS), MBPS (=10^6 BPS), GBPS (=10^9 BPS).

Options: R&S ZNB-K20

Manual operation: See "Data Rate" on page 407

CALCulate<Chn>:PULSe:INPut:PWIDth <PULSE>

Suffix:
<Chn> The channel number used to identify the active trace.

Parameters:
<PULSE> The width of the generated pulses.

Options: R&S ZNB-K20

Manual operation: See "Pulse Width" on page 407

CALCulate<Chn>:PULSe:INPut:SLICes <SLIC>

Suffix:
<Chn> The channel number used to identify the active trace.

Parameters:
<SLIC> specifies the number of slices, that is the samples per unit interval.

Options: R&S ZNB-K20

Manual operation: See "Slice (Samples/UI)" on page 407

CALCulate<Chn>:PULSe:MEASurement:CURSors:POST <PostCursor>

Defines the number of post cursors (right from the main cursor in the pulse response diagram).

Suffix:
<Chn> The channel number used to identify the active trace.

Parameters:
<PostCursor> The number of postcursors.

Options: R&S ZNB-K20

Manual operation: See "Cursors, Precursors, Post Cursors" on page 409

CALCulate<Chn>:PULSe:MEASurement:CURSors:PRE <PreCursor>

Defines the number of precursors (left from the main cursor in the pulse response diagram).

Suffix:
<Chn> The channel number used to identify the active trace.

Parameters:
<PreCursor> The number of precursors.

Options: R&S ZNB-K20

Manual operation: See "Cursors, Precursors, Post Cursors" on page 409

CALCulate<Chn>:PULSe:MEASurement:MAXTime <Max upper time>

Defines the horizontal range of the pulse response diagram. Only the signals within that range are displayed.

Suffix:
<Chn> The channel number used to identify the active trace.

Parameters:
<Max upper time> The maximum time defining the range of signals included in the pulse response measurement.

Options: R&S ZNB-K20

Manual operation: See "Time, Max Time" on page 409

CALCulate<Chn>:PULSe:MEASurement:STATe <Boolean>

If enabled, the pulse response diagram includes the main settings, cursor positions and the calculated ISI. The range of the pulse diagram can be configured, precursors and postcursors can be configured to be displayed.

Suffix:
<Chn> The channel number used to identify the active trace.

Parameters:
<Boolean> ON | OFF - enables or disables display measurement settings.

Options: R&S ZNB-K20

Manual operation: See "Display Measurements" on page 406

CALCulate<Chn>:PULSe:MEASurement:TYPE <ScpiPulseResponseMeasType>

Specifies display options of the pulse response diagram.

Suffix:
<Chn> The channel number used to identify the active trace.

Parameters:
<ScpiPulseResponseMeasType> FULL | TIME | CURSors

FULL: All signals are included in the measurement results, resulting in the maximum value of intersymbol interference (ISI).
TIME: the pulse response is displayed over the "Max Time" set with `CALCulate<Chn>:PULSe:MEASurement:MAXTime` on page 1080.
CURSors: Precursors and postcursors are displayed in the pulse response diagram. The horizontal range is adjusted for best representation of the cursors. The numbers of precursors and postcursors are defined by `CALCulate<Chn>:PULSe:MEASurement:CURSors:PRE` on page 1080 and `CALCulate<Chn>:PULSe:MEASurement:CURSors:POST` on page 1079.

Options: R&S ZNB-K20

Manual operation: See "Full" on page 409

CALCulate<Chn>:PULSe:STATe <Boolean>

Disables/enables pulse response measurements.

Suffix:
<Chn> The channel number used to identify the active trace.

Parameters:
<Boolean> ON | OFF - enables or disables pulse response measurements.

Options: R&S ZNB-K20

Manual operation: See "Pulse Response" on page 405

7.3.1.12 CALCulate:RIPPle...

The `CALCulate:RIPPle...` commands define the ripple limits and control the ripple limit check.

CALCulate<Chn>:RIPPle:CLEar....1082
CALCulate<Chn>:RIPPle:CONTrol:DOMain....1082
CALCulate<Chn>:RIPPle:DATA....1083
CALCulate<Chn>:RIPPle:DELete:ALL....1084
CALCulate:RIPPle:DISPlay:RESult:ALL[:STATe]....1084
CALCulate<Chn>:RIPPle:DISPlay[:STATe]....1084
CALCulate<Chn>:RIPPle:FAIL?....1085
CALCulate:RIPPle:FAIL:ALL?....1085

CALCulate<Chn>:RIPPle:RDOMain:FORMat....1085
CALCulate<Chn>:RIPPle:SEGMent:COUNt?....1086
CALCulate<Chn>:RIPPle:SEGMent<Seg>:LIMit....1086
CALCulate<Chn>:RIPPle:SEGMent<Seg>:RESult?....1087
CALCulate<Chn>:RIPPle:SEGMent<Seg>[:STATe]....1088
CALCulate<Chn>:RIPPle:SEGMent<Seg>:STIMulus:STARt....1088
CALCulate<Chn>:RIPPle:SEGMent<Seg>:STIMulus:STOP....1088
CALCulate<Chn>:RIPPle:SOUNd[:STATe]....1089
CALCulate<Chn>:RIPPle:STATe....1089
CALCulate<Chn>:RIPPle:STATe:AREA....1089

CALCulate<Chn>:RIPPle:CLEar

Resets the limit check results for the ripple test.

Suffix:
<Chn> Channel number used to identify the active trace

Usage: Event

Manual operation: See "Clear Test" on page 495

CALCulate<Chn>:RIPPle:CONTrol:DOMain <SweepType>

Deletes the existing ripple limit ranges and (re-)defines the physical units of the stimulus values of the ripple limit lines. The unit of the ripple limit is defined via `CALCulate<Chn>:RIPPle:RDOMain:FORMat`.

Suffix:
<Chn> Channel number used to identify the active trace

Setting parameters:
<SweepType> FLIN | FLOG | FSEG | FSINgle | TLIN | TLOG | PLIN | PLOG | PSINgle

Keywords for the units of the stimulus values; frequency, power, and time units.
The selected unit must be compatible with the sweep type (see `[SENSe<Ch>:]SWEep:TYPE` on page 1479): Hz for FLIN, FLOG, FSEG and FSINgle, s for TLIN and TLOG, dBm for PLIN, PLOG and PSINgle. Otherwise the ripple limit lines cannot be displayed and no ripple limit check is possible.

*RST: FLIN

Example:
```
SWE:TYPE POW
```
Select a power sweep.
```
CALC:RIPP:CONT:DOM PLIN
```
Delete all existing ripple limit ranges and select level units for the domain of the active trace.
```
CALC:RIPP:DATA 1, -10, -5, 3
```
Define and enable a ripple limit range in the stimulus range between -10 dBm and -5 dBm, assigning a ripple limit of 3 dB.

Usage: Setting only

Manual operation: See "Add / Insert / Delete / Delete All / Align All" on page 496

CALCulate<Chn>:RIPPle:DATA <RippleLimRange>...

Adds and enables/disables an arbitrary number of ripple limit ranges, assigning the stimulus values and the ripple limits. See Section 4.4.1.2, "Rules for ripple test definition", on page 138.

Note: This command does not overwrite existing ripple limit ranges. The defined ranges are appended to the range list as new ranges. Use the `CALCulate<Chn>:RIPPle:SEGMent<Seg>...` commands to change existing ripple limits.

Suffix:

<Chn> Channel number used to identify the active trace

Parameters:

<RippleLimRange> Parameter list in the format <Type>, <StartStimulus>, <StopStimulus>, <RippleLimit>[, {<Type>, <StartStimulus>, <StopStimulus>, <RippleLimit>}], where:
<Type> – Boolean identifier for the ripple limit range type. 1 for ripple limit range on (with limit check). 0 for ripple limit range off: The range is defined, but no limit check result displayed. The result is still available via `CALCulate<Chn>:RIPPle:SEGMent<Seg>:RESult?`.
<StartStimulus>/<StopStimulus>– stimulus values (unitless) confining the ripple limit range
<RippleLimit> – ripple limit (unitless) in the stimulus range between <StartStimulus> and <StopStimulus>
The unit of a stimulus value is adjusted to the sweep type of the active channel (`[SENSe<Ch>:]SWEep:TYPE`), the unit of a ripple limit is adjusted to the format of the active trace (`CALCulate<Chn>:FORMat`).

Range: Virtually no restriction for ripple limit ranges.
*RST: n/a (no ripple limit line defined after a reset)

Example:

```
*RST; CALC:RIPP:DATA 1, 1500000000, 2000000000,
3, 1, 2000000000, 3000000000, 5
```

Define and enable a ripple limit range in the stimulus range between 1.5 GHz and 2 GHz, assigning a ripple limit of +3 dB. Define and enable a second ripple limit range in the stimulus range between 2 GHz and 3 GHz, assigning a ripple limit of +5 dB.

```
CALC:RIPP:DISP ON
```

Show the ripple limits in the active diagram.

Manual operation: See "Add / Insert / Delete / Delete All / Align All" on page 496

CALCulate<Chn>:RIPPle:DELete:ALL

Deletes all ripple limit ranges.

Suffix:
<Chn> Channel number used to identify the active trace

Example: `*RST; CALC:RIPP:DATA 1,1500000000, 2000000000, 3, 1, 2000000000, 3000000000, 5`
Define and enable a ripple limit range in the stimulus range between 1.5 GHz and 2 GHz, assigning a ripple limit of +3 dB.
Define and enable a second ripple limit range in the stimulus range between 2 GHz and 3 GHz, assigning a ripple limit of +5 dB.
`CALC:RIPP:DEL:ALL`
Delete both created ripple limit ranges.

Usage: Event

Manual operation: See "Add / Insert / Delete / Delete All / Align All" on page 496

CALCulate:RIPPle:DISPlay:RESult:ALL[:STATe] <Enable>

Configures the display of ripple check info fields for the active recall set.

Parameters:
<Enable> ON - Info fields are displayed for all traces, for which a limit check is enabled.
OFF - Only the info field for the active trace is displayed (if the ripple check is enabled for this trace).
*RST: OFF

Manual operation: See "Show Results All Traces" on page 495

CALCulate<Chn>:RIPPle:DISPlay[:STATe] <Boolean>

Displays or hides all ripple limit lines (including all ranges) associated to the active trace.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<Boolean> ON | OFF - ripple limit line on or off.
*RST: OFF

Example: `*RST; CALC:RIPP:DATA 1,1500000000, 2000000000, 3`
Define and enable a ripple limit range in the stimulus range between 1.5 GHz and 2 GHz, assigning a ripple limit of +3 dB.
`CALC:RIPP:DISP ON`
Show the ripple limit range in the active diagram.

Manual operation: See "Show Ripple Limits" on page 493

CALCulate<Chn>:RIPPle:FAIL?

Returns a 0 or 1 to indicate whether or not the global ripple limit check has failed.

Tip: Use `CALCulate<Chn>:RIPPle:SEGMent<Seg>:RESult?` to query the result for a single ripple limit range.

Suffix:
<Chn> Channel number used to identify the active trace

Example:
```
*RST; CALC:RIPP:DATA 1, 1500000000, 2000000000, 3
```
Define and enable a ripple limit range in the stimulus range between 1.5 GHz and 2 GHz, assigning a ripple limit of +3 dB.
```
CALC:RIPP:STAT ON; FAIL?
```
Switch the limit check on and query the result.
```
CALC:RIPP:STAT:AREA LEFT, TOP
```
For a subsequent check at the GUI or a hardcopy, move the info field to the top left position.

Usage: Query only

Manual operation: See "Ripple Check" on page 494

CALCulate:RIPPle:FAIL:ALL? [<RecallSet>]

Returns a 0 or 1 to indicate whether or not the global ripple limit check has failed for at least one channel in the referenced recall set.

Query parameters:
<RecallSet> Recall set name; if omitted the active recall set is used

Usage: Query only

Manual operation: See "Ripple Check" on page 494

CALCulate<Chn>:RIPPle:RDOMain:FORMat <UnitRef>

Deletes the existing ripple limit ranges and (re-)defines the physical unit of the ripple limit. The units of the stimulus values are defined via `CALCulate<Chn>:RIPPle:CONTrol:DOMain`.

Suffix:
<Chn> Channel number used to identify the active trace

Setting parameters:

<UnitRef> COMPlex | MAGNitude | PHASe | REAL | IMAGinary | SWR | GDELay | L | C

Keyword for the physical unit of the response values; dimensionless numerss, relative power, phase, time, inductance, capacitance units.

*RST: n/a

Default unit: 1 (U, for COMPlex, REAL, IMAGinary, and SWR); dB (for MAGNitude), deg (for PHASe), s (for GDELay), H (Henry, for L), F (Farad, for C).

Example: `*RST; CALC:RIPP:DATA 1, 1500000000, 2000000000, 3`

Define and enable a ripple limit range in the stimulus range between 1.5 GHz and 2 GHz, assigning a ripple limit of +3 dB.

`CALC:RIPP:RDOM:FORM COMP`

Delete the ripple limit range, select complex units for the ripple limit.

Usage: Setting only

CALCulate<Chn>:RIPPle:SEGMent:COUNt?

Queries the number of ripple limit ranges. The response is an integer number.

Suffix:

<Chn> Channel number used to identify the active trace

Example: `*RST; CALC:RIPP:DATA 1, 1500000000, 2000000000, 3, 1, 2000000000, 3000000000, 5`

Define and enable a ripple limit range in the stimulus range between 1.5 GHz and 2 GHz, assigning a ripple limit of +3 dB. Define and enable a second ripple limit range in the stimulus range between 2 GHz and 3 GHz, assigning a ripple limit of +5 dB.

`CALC:RIPP:SEGM:COUNT?`

Query the number of ranges. The response is `2`.

Usage: Query only

Manual operation: See "Range List" on page 496

CALCulate<Chn>:RIPPle:SEGMent<Seg>:LIMit <Limit>

Defines the ripple limit for ripple limit range no. `<Seg>`. A range must be created first to enable this command (e.g. `CALCulate<Chn>:RIPPle:DATA`). See Section 4.4.1.2, "Rules for ripple test definition", on page 138.

Tip: To define several ripple limit ranges with a single command, use `CALCulate<Chn>:RIPPle:DATA`.

Suffix:

<Chn> Channel number used to identify the active trace

<Seg> Number of the ripple limit range.

Parameters:

<Limit> Ripple limit in the range. The unit is adjusted to the format of the active trace (`CALCulate<Chn>:FORMat`).

Range: Virtually no restriction for ripple limit ranges.
*RST: n/a (no ripple limit line defined after a reset)
Default unit: See above.

Example: See `CALCulate<Chn>:RIPPle:SEGMent<Seg>:STIMulus:STARt`

Manual operation: See "Range List" on page 496

CALCulate<Chn>:RIPPle:SEGMent<Seg>:RESult?

Returns the result of the ripple limit check in the previously defined limit range no. `<Seg>`. The response consists of two parameters:

- <Boolean> – 0 for "passed", 1 for "failed".
- <Limit> – measured ripple in the limit range. A result is returned even if the limit check in the range no. <Seg> is disabled; see example below.

A reset deletes all ripple limit ranges. Use `CALCulate<Ch>:RIPPle:FAIL?` to query the result for global ripple limit check.

Note: In remote control, the ripple limit check result is calculated once at the end of each sweep. If the ripple limits are changed, a new sweep is required to obtain updated ripple limit check results. In single sweep mode (`INITiate<Ch>:CONTinuous OFF`), the new sweep must be started explicitly. This behavior is different from manual control where a changed ripple limit line can directly affect the pass/fail result of the displayed trace.

Suffix:

<Chn> Channel number used to identify the active trace

<Seg> Number of the ripple limit range.

Example:

```
*RST; CALC:RIPP:DATA 1, 1500000000, 2000000000, 3
```

Define and enable a ripple limit range in the stimulus range between 1.5 GHz and 2 GHz, assigning a ripple limit of +3 dB.

```
CALC:RIPP:STAT ON; SEGM:RES?
```

Enable the limit check and query the result for the created range. Possible response: `0,0.3529814004`.

```
CALC:RIPP:DATA 0, 2500000000, 3000000000, 3
```

Define a second ripple limit range with disabled limit check (no limit check results are displayed in the diagram area).

```
CALC:RIPP:SEGM2:RES?
```

Query the result for the second range. Possible response: `0,1.149071925`.

Usage: Query only

Manual operation: See "Ripple Check" on page 494

CALCulate<Chn>:RIPPle:SEGMent<Seg>[:STATe] <Boolean>

Enables or disables the limit check in the ripple limit range no. <Seg>.

Suffix:
<Chn> Channel number used to identify the active trace

<Seg> Number of the ripple limit range.

Parameters:
<Boolean> ON | OFF - Limit check on or off. A result is available even if the limit check is disabled; see example for CALCulate<Chn>:RIPPle:SEGMent<Seg>:RESult?.

*RST: n/a (no ripple limit line defined after a reset)

Example: See CALCulate<Chn>:RIPPle:SEGMent<Seg>:STIMulus:STARt

Manual operation: See "Range List" on page 496

CALCulate<Chn>:RIPPle:SEGMent<Seg>:STIMulus:STARt <FreqPowTime>
CALCulate<Chn>:RIPPle:SEGMent<Seg>:STIMulus:STOP <StimValue>

These commands change the start or stop stimulus values (i.e. the smallest or largest stimulus values) of a ripple limit range. A range must be created first to enable these commands (e.g CALCulate<Chn>:RIPPle:DATA). See Section 4.4.1.2, "Rules for ripple test definition", on page 138.

Tip: To define several ripple limit ranges with a single command, use CALCulate<Chn>:RIPPle:DATA.

Suffix:
<Chn> Channel number used to identify the active trace

<Seg> Number of the ripple limit range.

Parameters:
<StimValue> Stimulus values (unitless) confining the ripple limit range.
The unit is adjusted to the sweep type of the active channel ([SENSe<Ch>:]SWEep:TYPE).

Range: Virtually no restriction for ripple limit ranges.
*RST: n/a (no ripple limit line defined after a reset)
Default unit: NN

Example: `*RST; CALC:RIPP:DATA 1,1500000000, 2000000000,3`
Define and enable a ripple limit range in the stimulus range between 1.5 GHz and 2 GHz, assigning a ripple limit of +3 dB.
`CALC:RIPP:SEGM:STIM:STAR 1GHZ; STOP 2.5 GHZ; :`
`CALC:RIPP:SEGM:LIM 5`
Change the range to a stimulus range between 1 GHz and 2.5 GHz and a limit of 5 dB.
`CALC:RIPP:SEGM:STAT OFF`
Disable the limit check in the modified stimulus range.

Manual operation: See "Range List" on page 496

CALCulate<Chn>:RIPPle:SOUNd[:STATe] <Boolean>

Deprecated command: Since version 3.45, generating sounds is no longer supported by the analyzer firmware.

Switches the acoustic signal (fail beep) on or off. The fail beep is generated each time the analyzer detects an exceeded ripple limit.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<Boolean> ON | OFF - fail beep on or off.
*RST: OFF

Example: `CALC:RIPP:STAT ON; SOUN ON`
Switch on the limit check and activate the fail beep.

CALCulate<Chn>:RIPPle:STATe <Boolean>

Switches the ripple limit check for the active trace on or off.

Tip: Use `CALCulate<Chn>:RIPPle:SEGMent<Seg>[:STATe]` to switch the limit check for a single ripple limit range on or off.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<Boolean> ON | OFF – ripple limit check on or off.
*RST: OFF

Example: See `CALCulate<Chn>:RIPPle:FAIL?`

Manual operation: See "Ripple Check" on page 494

CALCulate<Chn>:RIPPle:STATe:AREA <HorizontalPos>, <VerticalPos>

Moves the ripple test info field for the active trace <Chn> to one of nine predefined positions in the active diagram.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<HorizontalPos> LEFT | MID | RIGHt
Horizontal position

<VerticalPos> TOP | MID | BOTTom
Vertical position

Example: See `CALCulate<Chn>:RIPPle:FAIL?`

Manual operation: See "Ripple Check" on page 494

7.3.1.13 CALCulate:STATistics...

The `CALCulate:STATistics...` commands evaluate and display statistical and phase information of the trace.

CALCulate<Chn>:STATistics:DOMain:USER ... 1090
CALCulate<Chn>:STATistics:DOMain:USER:SHOW ... 1091
CALCulate<Chn>:STATistics:DOMain:USER:STARt ... 1091
CALCulate<Chn>:STATistics:DOMain:USER:STOP ... 1091
CALCulate<Chn>:STATistics:EPDelay[:STATe] ... 1092
CALCulate<Chn>:STATistics:MMPTpeak[:STATe] ... 1092
CALCulate<Chn>:STATistics:MSTDdev[:STATe] ... 1092
CALCulate<Chn>:STATistics:FORMat ... 1092
CALCulate<Chn>:STATistics:NLINear:COMP:LEVel ... 1092
CALCulate<Chn>:STATistics:NLINear:COMP:PHASe ... 1093
CALCulate<Chn>:STATistics:NLINear:COMP:RDOMain:USER ... 1093
CALCulate<Chn>:STATistics:NLINear:COMP:RDOMain:USER:SHOW ... 1093
CALCulate<Chn>:STATistics:NLINear:COMP:RDOMain:USER:STARt ... 1094
CALCulate<Chn>:STATistics:NLINear:COMP:RDOMain:USER:STOP ... 1094
CALCulate<Chn>:STATistics:NLINear:COMP:REFerence ... 1094
CALCulate<Chn>:STATistics:NLINear:COMP:RESult? ... 1094
CALCulate<Chn>:STATistics:NLINear:COMP:RLEVel ... 1095
CALCulate<Chn>:STATistics:NLINear:COMP:RMARker ... 1096
CALCulate<Chn>:STATistics:NLINear:COMP:RPHase ... 1096
CALCulate<Chn>:STATistics:NLINear:COMP[:STATe] ... 1096
CALCulate<Chn>:STATistics:RESult? ... 1097
CALCulate<Chn>:STATistics:RMS[:STATe] ... 1098
CALCulate<Chn>:STATistics:SFLatness[:STATe] ... 1098
CALCulate<Chn>:STATistics[:STATe] ... 1098
CALCulate<Chn>:STATistics[:STATe]:AREA ... 1099

CALCulate<Chn>:STATistics:DOMain:USER <EvalRange>

Selects one out of 10 evaluation ranges to be configured with the `CALCulate<Chn>:STATistics:DOMain:USER:SHOW`, `CALCulate<Chn>:STATistics:DOMain:USER:STARt`, and `CALCulate<Chn>:STATistics:DOMain:USER:STOP` commands.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<EvalRange> Number of the evaluation range.
Range: 1 to 10. In addition, 0 denotes the (non-configurable) "Full Span" evaluation range.
*RST: 0

Example: `*RST; :CALC:STAT:DOM:USER?`
Query the default evaluation range. The response is zero, i.e. the evaluation range is equal to the complete sweep range
`CALC:STAT:DOM:USER 1`
`CALC:STAT:DOM:USER:STARt 1GHZ; STOP 2GHZ; SHOW ON`
Select evaluation range no. 1 and define the evaluation range between 1 GHz and 2 GHz. Display the range limit lines.

Manual operation: See "Evaluation Range" on page 470

CALCulate<Chn>:STATistics:DOMain:USER:SHOW <Boolean>

Displays or hides range limit lines for the evaluation range selected via `CALCulate<Chn>:STATistics:DOMain:USER`.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<Boolean> ON | OFF - range limit lines on or off.
*RST: OFF

Example: See `CALCulate<Chn>:STATistics:DOMain:USER`

Manual operation: See "Range Limit Lines On" on page 471

CALCulate<Chn>:STATistics:DOMain:USER:STARt <Start>
CALCulate<Chn>:STATistics:DOMain:USER:STOP <Stop>

These commands define the start and stop values of the evaluation range selected via `CALCulate<Chn>:STATistics:DOMain:USER`.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<Stop> Start or stop value of the evaluation range.
Default unit: NN

Example: See `CALCulate<Chn>:STATistics:DOMain:USER`

Manual operation: See "Evaluation Range" on page 470

CALCulate<Chn>:STATistics:EPDelay[:STATe] <Boolean>
CALCulate<Chn>:STATistics:MMPTpeak[:STATe] <Boolean>
CALCulate<Chn>:STATistics:MSTDdev[:STATe] <Boolean>

These commands display or hide the "Phase/El Length" results, the "Min/Max/Peak-Peak" results, and the "Mean/Std Dev" results in the diagram area of trace no. <Chn>.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<Boolean> ON | OFF - statistical info field on or off.
*RST: OFF

Example: See CALCulate<Chn>:STATistics[:STATe]

Manual operation: See "Min/Max/Peak-Peak, Mean/Std Dev/RMS" on page 466

CALCulate<Chn>:STATistics:FORMat <Format>

For complex-valued traces (Smith, Polar) this determines how the MEAN, STDDev, MAX, MIN, RMS and PTPeak statistics are calculated, see CALCulate<Chn>:STATistics:RESult?.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<Format> ZVAB | IMPedance | ADMittance

ZVAB
The results are based on unformatted wave quantities (voltages)

IMPedance
The results are based on resistance values

ADMittance
The results are based on conductance vaules

Manual operation: See "Format" on page 467

CALCulate<Chn>:STATistics:NLINear:COMP:LEVel <dBValue>

Defines the compression value x for the compression point measurement for dB formatted traces.

Use CALCulate<Chn>:STATistics:NLINear:COMP:RESult? to retrieve the compression results.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
dBValue Compression value
Default unit: dB

Example: See `CALCulate<Chn>:STATistics:NLINear:COMP:RESult?`

Manual operation: See "Compr. Point / Compr. Val." on page 468

CALCulate<Chn>:STATistics:NLINear:COMP:PHASe <PhaseValue>

Defines the compression value x for the compression point measurement for phase formatted traces.

Use `CALCulate<Chn>:STATistics:NLINear:COMP:RESult?` to retrieve the compression results.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<PhaseValue> Compression value
Default unit: deg

Manual operation: See "Compr. Point / Compr. Val." on page 468

CALCulate<Chn>:STATistics:NLINear:COMP:RDOMain:USER <RefRange>

Selects one of the 10 channel-specific user defined ranges as the reference range for compression point calculation.

Only applies if `CALCulate<Chn>:STATistics:NLINear:COMP:REFerence` is set to `RANGe`.

The selected range can then be configured with the `CALCulate<Chn>:STATistics:NLINear:COMP:RDOMain:USER:STARt`, `CALCulate<Chn>:STATistics:NLINear:COMP:RDOMain:USER:STOP`, and `CALCulate<Chn>:STATistics:NLINear:COMP:RDOMain:USER:SHOW` commands.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<RefRange> Range number
Range: 1 to 10. In addition, 0 denotes the (non-configurable) "Full Span" evaluation range.
*RST: 0

Manual operation: See "Ref. Range" on page 469

CALCulate<Chn>:STATistics:NLINear:COMP:RDOMain:USER:SHOW <Boolean>

Displays or hides range limit lines for the reference range selected via `CALCulate<Chn>:STATistics:NLINear:COMP:RDOMain:USER`.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<Boolean> ON | OFF – reference range limit lines on or off.
*RST: OFF

Manual operation: See "Ref. Range" on page 469

CALCulate<Chn>:STATistics:NLINear:COMP:RDOMain:USER:STARt <Start>
CALCulate<Chn>:STATistics:NLINear:COMP:RDOMain:USER:STOP <Value>

Defines the start/stop value of the range selected via `CALCulate<Chn>:STATistics:NLINear:COMP:RDOMain:USER`.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<Value> Start/stop value of the related range

Manual operation: See "Ref. Range" on page 469

CALCulate<Chn>:STATistics:NLINear:COMP:REFerence <ReferenceValue>

Defines how the reference value ("small signal value") for the compression point calculation is calculated.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<ReferenceValue> FPOint | DVALue | MARKer | RANGe

FPOint – uses the value at the first point of the selected evaluation range as the reference value (see `CALCulate<Chn>:STATistics:DOMain:USER`)
DVALue – allows to specify the reference value manually (see `CALCulate<Chn>:STATistics:NLINear:COMP:RLEVel` and `CALCulate<Chn>:STATistics:NLINear:COMP:RPHase`)
MARKer: uses the value at a selectable marker as the reference value (see `CALCulate<Chn>:STATistics:NLINear:COMP:RMARker`)
RANGe – uses the average value in a configurable reference range as the reference value (see `CALCulate<Chn>:STATistics:NLINear:COMP:RDOMain:USER`)

*RST: FPOint

Manual operation: See "Reference Value" on page 469

CALCulate<Chn>:STATistics:NLINear:COMP:RESult?

Returns the compression point of an S-parameter or ratio measured in a power sweep.

For dB formatted traces, the compression value x is set via `CALCulate<Chn>:STATistics:NLINear:COMP:LEVel`, for phase formatted traces it is set via `CALCulate<Chn>:STATistics:NLINear:COMP:PHASe`.

The response contains two numeric values:

- <Cmp In> – stimulus level at the compression point in dBm.
- <Cmp Out>
  - For dB formatted traces, the sum of <Cmp In> plus the magnitude of the measured response value at the compression point in dBm.
  - For phase formatted traces, the response value at the compression point

Suffix:
<Chn> Channel number used to identify the active trace

Example:
```
*RST; SWE:TYPE POW
```
Select a power sweep with default CW frequency and sweep range.
```
CALC:STAT:NLIN:COMP:LEV 2
```
Define a compression value of 2 dB.
```
CALC:STAT:NLIN:COMP:RES?
```
Query the compression point results <Cmp In>, <Cmp Out>. An execution error message (error no. -200) is returned if no compression point is found.
```
CALC:STAT:NLIN:COMP ON
```
Display the compression point result in the diagram area.

Usage: Query only

Manual operation: See "Compr. Point / Compr. Val." on page 468

CALCulate<Chn>:STATistics:NLINear:COMP:RLEVel <Level>

Manually defines the reference level for compression point calculation on dB formatted traces.

This only applies if `CALCulate<Chn>:STATistics:NLINear:COMP:REFerence` is set to `DVALue`. For phase formatted traces, the reference level can be set using `CALCulate<Chn>:STATistics:NLINear:COMP:RPHase`.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<Level> Reference level

*RST: 1 dB
Default unit: dB

Manual operation: See "Defined Value" on page 470

CALCulate<Chn>:STATistics:NLINear:COMP:RMARker <Marker>

Allows you to select the marker whose value shall be used as the reference ("small signal value") for the compression point calculation.

Only applies if `CALCulate<Chn>:STATistics:NLINear:COMP:REFerence` is set to `MARKer`.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<Marker> Marker number
If marker <Marker> is OFF for the related trace, the set command is silently ignored.

Manual operation: See "Selected Marker" on page 469

CALCulate<Chn>:STATistics:NLINear:COMP:RPHase <Phase>

Manually defines the reference phase for compression point calculation on phase formatted traces.

Only applies if `CALCulate<Chn>:STATistics:NLINear:COMP:REFerence` is set to `DVALue`.

For dB formatted traces, the reference level can be set using `CALCulate<Chn>:STATistics:NLINear:COMP:RLEVel`.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<Phase> Reference phase
*RST: 1°
Default unit: deg

Manual operation: See "Defined Value" on page 470

CALCulate<Chn>:STATistics:NLINear:COMP[:STATe] <Boolean>

Displays or hides the compression point result in the diagram area channel `<Chn>`'s active trace..

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<Boolean> ON | OFF - statistical info field on or off.
*RST: OFF

Example: See `CALCulate<Chn>:STATistics:NLINear:COMP:RESult?`

Manual operation: See "Compr. Point / Compr. Val." on page 468

CALCulate<Chn>:STATistics:RESult? <Result>

Returns a single statistical parameter of the trace no. <Chn> or all parameters. It is not necessary to display the info field (CALCulate<Chn>:STATistics[:STATe] ON) before using this command.

Suffix:
<Chn> Channel number used to identify the active trace

Query parameters:
<Result> MEAN | STDDev | MAX | MIN | RMS | PTPeak | PEAK2p | ELENgth | PDELay | GAIN | SLOPe | FLATness | ALL

MEAN - return arithmetic mean value of all response values of the trace in the entire sweep range (or in the evaluation range defined in manual control).
STDDev - return standard deviation of all response values.
MAX - return the maximum of all response values.
MIN - return the minimum of all response values.
RMS - return the root mean square of all response values.
PTPeak - return the peak-to-peak value (MAX - MIN).
ELENgth - return the electrical length.
PDELay - return the phase delay.
GAIN - return the gain, i.e. the larger of two marker values.
SLOPe - return the slope (difference) between two marker values.
FLATness - return the flatness of the trace between two marker positions.
ALL - return all statistical values, observing the order used above.
The data is returned as a comma-separated list of real numbers. The unit is the default unit of the measured parameter (see CALCulate<Ch>:PARameter:SDEFine) but may also depend on the trace format (see CALCulate<Chn>:FORMat). For complex traces the statistical results MEAN, STDDev, MAX, MIN, RMS and PTPeak are calculated in the selected format (see CALCulate<Chn>:STATistics:FORMat).

Example:
```
*RST; :CALC:STAT:RES? MAX
```
Calculate and return the maximum of the default trace showing an S-parameter on a dB Mag scale.
```
:CALC:FORM POL; STAT:RES? MAX
```
Display the trace in a polar diagram and re-calculate the maximum. The result corresponds to the previous result but is converted to a unitless linear value.

Usage: Query only

Manual operation: See "Min/Max/Peak-Peak, Mean/Std Dev/RMS" on page 466

CALCulate<Chn>:STATistics:RMS[:STATe] <Boolean>
CALCulate<Chn>:STATistics:SFLatness[:STATe] <Boolean>

These commands display or hide the "RMS" and the "Flatness/Gain/Slope" results in the diagram area of trace no. <Chn>.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<Boolean> ON | OFF - statistical info field on or off.
*RST: OFF

Example: See CALCulate<Chn>:STATistics[:STATe]

Manual operation: See "Flatness/Gain/Slope" on page 467

CALCulate<Chn>:STATistics[:STATe] <Boolean>

Displays or hides all statistical results in the diagram area of trace no. <Chn> except the compression point results.

Tip: You can display or hide the "Min/Max/Peak-Peak", "Mean/Std Dev/RMS", "Phase/El Length" and "Flatness/Gain/Slope" results separately; see example below.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<Boolean> ON | OFF - Statistical info field on or off.
*RST: OFF

Example:
```
*RST; :CALC:STAT:MMPT ON
```
Reset the instrument, hiding all statistical results. Display the "Min/Max/Peak-Peak" results.
```
CALC:STAT:MSTD ON
```
Display the "Mean/Std Dev" results in addition.
```
CALC:STAT:RMS ON
```
Display the "RMS" results in addition.
```
CALC:STAT:EPD ON
```
Display the "Phase/El Length" results in addition.
```
CALC:STAT:SFL ON
```
Display the "Flatness/Gain/Slope" results in addition.
```
CALC:STAT:STAT:AREA LEFT, TOP
```
For a subsequent check at the GUI or a hardcopy, move the info field to the top left position.
...
```
CALC:STAT OFF
```
Hide all results.

Manual operation: See "Min/Max/Peak-Peak, Mean/Std Dev/RMS" on page 466

CALCulate<Chn>:STATistics[:STATe]:AREA <HorizontalPos>, <VerticalPos>

Moves the statistics info field for the active trace <Chn> to one of nine predefined positions in the active diagram.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<HorizontalPos> LEFT | MID | RIGHt
Horizontal position

<VerticalPos> TOP | MID | BOTTom
Vertical position

Example: See `CALCulate<Chn>:STATistics[:STATe]`

Manual operation: See "Min/Max/Peak-Peak, Mean/Std Dev/RMS" on page 466

7.3.1.14 CALCulate:TRANsform...

The `CALCulate:TRANsform...` commands convert measured data from one representation to another.

- CALCulate:TRANsform:DTFault... 1099
- CALCulate:TRANsform:TIME... 1105
- CALCulate:TRANsform:VNETworks... 1114
- CALCulate:TRANsform... (other) 1164

CALCulate:TRANsform:DTFault...

The `CALCulate:TRANsform:DTFault...` commands set up and control a distance to fault measurement (with option R&S ZNB-K3).

CALCulate<Chn>:TRANsform:DTFault:CENTer 1100
CALCulate<Chn>:TRANsform:DTFault:SPAN 1100
CALCulate<Chn>:TRANsform:DTFault:DEFine 1100
CALCulate<Chn>:TRANsform:DTFault:DELete 1101
CALCulate:TRANsform:DTFault:EXPort 1101
CALCulate<Chn>:TRANsform:DTFault:PEAK:COUNt? 1102
CALCulate<Chn>:TRANsform:DTFault:PEAK:DATA<FaultNo> 1102
CALCulate<Chn>:TRANsform:DTFault:PEAK:STATe 1103
CALCulate<Chn>:TRANsform:DTFault:PEAK:THReshold 1103
CALCulate<Chn>:TRANsform:DTFault:POINts 1104
CALCulate<Chn>:TRANsform:DTFault:SELect 1104
CALCulate<Chn>:TRANsform:DTFault:STARt 1105
CALCulate<Chn>:TRANsform:DTFault:STOP 1105
CALCulate<Chn>:TRANsform:DTFault:STATe 1105

CALCulate<Chn>:TRANsform:DTFault:CENTer <Center>
CALCulate<Chn>:TRANsform:DTFault:SPAN <Span>

Defines the distance window of the Distance to Fault measurement using its center and span.

See also `CALCulate<Chn>:TRANsform:DTFault:STARt` and `CALCulate<Chn>:TRANsform:DTFault:STOP`.

Suffix:
<Chn> Channel number used to identify the active trace.

Parameters:
<Center>/<Span> Center/span value.
Default unit: m

Example: `:CALCulate1:TRANsform:DTFault:CENTer 2; SPAN 2`
Defines the DtF distance window of the active trace of channel 1 via its center (2m) and span (2m).

Options: R&S ZNB-K3

Manual operation: See "Start Distance / Stop Distance" on page 367

CALCulate<Chn>:TRANsform:DTFault:DEFine <CblName>, <CblPermittivity>, <CblAtt1>, <CblFreq1>{, <CblAtt2>, <CblFreq2>}...

Defines a new user-defined cable type for Distance to Fault (DtF) measurements.

The cable's frequency-dependent attenuation is defined via attenuation/frequency pairs `<CblAtt1>, <CblFreq1>, <CblAtt2>, <CblFreq2>, ...`. At least one pair has to be specified.

Suffix:
<Chn> Channel number
This suffix is ignored: cable types are defined for all channels.

Setting parameters:
<CblName> Name of the cable type.
Must be unique among all DtF cable types, with case insensitive match.

<CblPermittivity> Relative permittivity ε_r of the cable type.

<CblAtt[i]> Attenuation value *i* in the frequency-dependent attenuation table of the cable type.
Default unit: dB/m

<CblFreq[i]> Frequency *i* in the frequency-dependent attenuation table of the cable type.
Default unit: Hz

Example: `CALCulate:TRANsform:DTFault:DEFine 'My cable type', 1.1, 0.01, 1GHz, 0.015, 2GHz, 0.012, 3GHz`
Defines a new cable type "My cable type" with relative permittivity ε_r=1.1 and the frequency-dependent attenuation given in the table below.

Usage: Setting only

Options: R&S ZNB-K3

Manual operation: See "Add / Delete" on page 372

Table 7-4: Example: frequency-dependent attenuation table

[i]	Frequency	Attenuation
1	1 GHz	0.01 dB/m
2	2 GHz	0.015 dB/m
3	3 GHz	0.012 dB/m

CALCulate<Chn>:TRANsform:DTFault:DELete <DtfDeleteCable>

Deletes the user-defined cable type with the given name.

Suffix:
<Chn> Channel number
This suffix is ignored: cable types are defined for (and deleted from) all channels.

Setting parameters:
<DtfDeleteCable> Name of a user-defined cable type.

Example: `CALCulate:TRANsform:DTFault:DELete 'My cable type'`
Deletes the user-defined cable type "My cable type".

Usage: Setting only

Options: R&S ZNB-K3

Manual operation: See "Add / Delete" on page 372

CALCulate:TRANsform:DTFault:EXPort <ExportFile>[, <Contents>[, <DecSeparator>[, <FieldSeparator>]]]

Exports the fault list of a distance-to-fault measurement to an ASCII file (`*.csv`).

Setting parameters:
<ExportFile> File name or path

<Contents> ACTTrace | ACTChannel | ALLChannels

ACTTrace
Active trace

ACTTrace
Active channel

ALLChannels
All channels

<DecSeparator> POINt | COMMa
Decimal separator

<FieldSeparator> SEMicolon | COMMa | TABulator | SPACe
Field separator

Usage: Setting only

Options: R&S ZNB-K3

CALCulate<Chn>:TRANsform:DTFault:PEAK:COUNt?

If the active trace of channel <Chn> is a Distance to Fault (DtF) trace and DtF limit checking is enabled, this query returns the number of DtF limit violations of this trace.

Use `CALCulate<Chn>:TRANsform:DTFault:PEAK:STATe` to enable DtF limit checking and `CALCulate<Chn>:TRANsform:DTFault:PEAK:THReshold` to set the fault limit.

Suffix:
<Chn> Channel number used to identify the active trace

Usage: Query only

Options: R&S ZNB-K3

Manual operation: See "Fault Table" on page 373

CALCulate<Chn>:TRANsform:DTFault:PEAK:DATA<FaultNo>

If the active trace of channel <Chn> is a Distance to Fault (DtF) trace and DtF limit checking is enabled, this query returns the peak data of DtF limit violation <FaultNo>

<FaultNo> must be between 1 and `CALCulate<Chn>:TRANsform:DTFault:PEAK:COUNt?`. The peak data are returned as value pair `distance, response value`. Peaks are numbered according to their distance (fault no. 1 has the smallest distance).

Use `CALCulate<Chn>:TRANsform:DTFault:PEAK:STATe` to enable DtF limit checking and `CALCulate<Chn>:TRANsform:DTFault:PEAK:THReshold` to set the fault limit.

Suffix:
<Chn> Channel number used to identify the active trace

<FaultNo> 1 ... CALCulate<Chn>:TRANsform:DTFault:PEAK:COUNt?

Example: Suppose `CALCulate1:TRANsform:DTFault:PEAK:COUNt?` returns 3, then
`CALCulate1:TRANsform:DTFault:PEAK:DATA1; DATA2; DATA3`
returns the coordinates of the DtF limit violation peaks.

Options: R&S ZNB-K3

Manual operation: See "Fault Table" on page 373

CALCulate<Chn>:TRANsform:DTFault:PEAK:STATe <DtfPeakState>

If the active trace of channel <Chn> is a Distance to Fault (DtF) trace, this command allows to enable/disable DtF limit checking.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<DtfPeakState> OFF|ON (0|1)

Example: `*RST; :CALCulate1:TRANsform:DTFault:STATe ON`
makes the active trace of channel 1 a DtF trace.
`CALCulate1:TRANsform:DTFault:PEAK:STATe ON`
enables DtF limit checking.

Options: R&S ZNB-K3

Manual operation: See "Fault Limit Check" on page 369

CALCulate<Chn>:TRANsform:DTFault:PEAK:THReshold <PeakThreshold>

If the active trace of channel <Chn> is a Distance to Fault (DtF) trace, this command allows to set the fault limit for DtF limit checking.

Use `CALCulate<Chn>:TRANsform:DTFault:PEAK:STATe` to enable DtF limit checking.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<PeakThreshold> Fault limit

*RST: -20 dB
Default unit: dB

Example: `*RST; :CALCulate1:TRANsform:DTFault:STATe ON`
makes the active trace of channel 1 a DtF trace.
`CALCulate1:TRANsform:DTFault:PEAK:THReshold -30`
sets the DtF limit to -30 dB.

Options: R&S ZNB-K3

Manual operation: See "Fault Limit" on page 370

CALCulate<Chn>:TRANsform:DTFault:POINts

If the active trace of channel <Chn> is a Distance to Fault (DtF) trace, this command starts the "Auto Number of Points" calculation (see "Auto Number of Points" on page 368).

An execution error is raised, if the calculated number of points is higher than the maximum number of sweep points N_{max} = 100001 the firmware allows. To avoid this, reduce the frequency span Δf (see `[SENSe<Ch>:]SWEep:POINts` on page 1475) and/or the stop frequency d_{stop} (see `CALCulate<Chn>:TRANsform:DTFault:STOP`) so that

$2.6 \cdot d_{stop} \cdot \Delta f / (v \cdot c_0) \leq N_{max}$,

where *v* denotes the velocity factor of the measured cable (see `CALCulate<Chn>:TRANsform:DTFault:SELect`).

Suffix:
<Chn> Channel number used to identify the active trace

Example:
```
*RST; :CALCulate1:TRANsform:DTFault:STATe ON
```
makes the active trace of channel 1 a DtF trace.
```
SENSe1:SWEep:POINts?
```
returns the default number of sweep points (201).
```
:CALCulate1:TRANsform:DTFault:STARt 0; STOP 100
```
widens the DtF distance window to 100m.
```
CALCulate1:TRANsform:DTFault:POINts
```
starts the "Auto Number of Points calculation". Now
```
SENSe1:SWEep:POINts?
```
returns an increased number of sweep points.

Usage: Event

Options: R&S ZNB-K3

Manual operation: See "Auto Number of Points" on page 368

CALCulate<Chn>:TRANsform:DTFault:SELect <DtfSelectCable>

Selects one of the available (predefined or user-defined) cable types for Distance to Fault measurements.

Suffix:
<Chn> Channel number used to identify the active trace.

Parameters:
<DtfSelectCable> Name of the cable type.

Example:
```
CALCulate1:TRANsform:DTFault:SELect '5088-HFLR'
```
Selects cable type "5088-HFLR".

Options: R&S ZNB-K3

Manual operation: See "Cable Type" on page 368

CALCulate<Chn>:TRANsform:DTFault:STARt <Start>
CALCulate<Chn>:TRANsform:DTFault:STOP <Stop>

Defines the window of the Distance to Fault measurement using its start and stop distance.

See also `CALCulate<Chn>:TRANsform:DTFault:CENTer` and `CALCulate<Chn>:TRANsform:DTFault:SPAN`.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<Start>/<Stop> Start/stop distance.
Default unit: m

Example: `:CALCulate1:TRANsform:DTFault:STARt 1; STOP 3`
Defines the DtF distance window of the active trace of channel 1 as [1m, 3m].

Options: R&S ZNB-K3

Manual operation: See "Start Distance / Stop Distance" on page 367

CALCulate<Chn>:TRANsform:DTFault:STATe <DtfState>

If channel <Chn> is performing an unsegmented frequency sweep, this command allows to transform its active trace into a Distance to Fault (DtF) trace.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<DtfState> OFF|ON – DtF disabled|enabled
*RST: OFF

Example: `:CALCulate1:TRANsform:DTFault:STATE ON`
turns the DtF representation of the active trace of channel 1 ON.

Options: R&S ZNB-K3

Manual operation: See "Distance to Fault" on page 367

CALCulate:TRANsform:TIME...

The `CALCulate:TRANsform:TIME...` commands control the transformation into the time domain (see Section 4.7.1, "Time domain analysis", on page 218).

CALCulate<Chn>:TRANsform:TIME:CAUSal 1106
CALCulate<Chn>:TRANsform:TIME:CENTer 1106
CALCulate<Chn>:TRANsform:TIME:DCHebyshev 1107
CALCulate<Chn>:TRANsform:TIME:EXPonential 1107
CALCulate<Chn>:TRANsform:TIME:GAUSsian 1107
CALCulate<Chn>:TRANsform:TIME:KAISer 1108
CALCulate<Chn>:TRANsform:TIME:LPASs 1108

CALCulate<Chn>:TRANsform:TIME:LPASs:DCSParam 1109
CALCulate<Chn>:TRANsform:TIME:LPASs:DCSParam:CONTinuous 1109
CALCulate<Chn>:TRANsform:TIME:LPASs:DCSParam:EXTRapolate 1110
CALCulate<Chn>:TRANsform:TIME:LPFRequency 1110
CALCulate<Chn>:TRANsform:TIME:RESolution:EFACtor 1110
CALCulate<Chn>:TRANsform:TIME:SPAN 1111
CALCulate<Chn>:TRANsform:TIME:STARt 1111
CALCulate<Chn>:TRANsform:TIME:STATe 1111
CALCulate<Chn>:TRANsform:TIME:STIMulus 1112
CALCulate<Chn>:TRANsform:TIME:STOP 1112
CALCulate<Chn>:TRANsform:TIME[:TYPE] 1113
CALCulate<Chn>:TRANsform:TIME:WINDow 1113
CALCulate<Chn>:TRANsform:TIME:XAXis 1114

CALCulate<Chn>:TRANsform:TIME:CAUSal <RiseTime>

Sets the rise time for the causal rise time window.

This command is only available if the causal window is active (`CALCulate<Chn>:TRANsform:TIME:WINDow CAUSal`).

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<RiseTime> Configurable rise time
Default unit: s

Options: R&S ZNB-K2

Manual operation: See "Causal Rise Time" on page 360

CALCulate<Chn>:TRANsform:TIME:CENTer <CenterTime>

Defines the center time of the diagram in time domain.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<CenterTime> Center time of the diagram in time domain.
Range: -99.99999999999 s to +99.99999999999 s
Increment: 0.1 ns
*RST: 1.5E-009 s
Default unit: s

Example:
```
*RST; :CALC:TRAN:TIME:STAT ON
```
Reset the instrument, activating a frequency sweep, and enable the time domain transformation for the default trace.
```
CALC:TRAN:TIME:CENT 0; SPAN 5ns
```
Set the center time to 0 ns and the time span to 5 ns.

Manual operation: See "Time Start / Time Stop / Time Center / Time Span" on page 529

Note: If the x-axis is scaled in distance units (`CALCulate<Chn>:TRANsform:TIME:XAXis DISTance`), then the center value is entered in m; the range and default value changes accordingly.

CALCulate<Chn>:TRANsform:TIME:DCHebyshev <SidebandSupp>

Sets the sideband suppression for the Dolph-Chebyshev window. The command is only available if a Dolph-Chebyshev window is active (`CALCulate<Chn>:TRANsform:TIME:WINDow DCHebyshev`).

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<SidebandSupp> Sideband suppression

Range: 10 dB to 120 dB
Increment: 10 dB
*RST: 32 dB
Default unit: dB

Example: `*RST; :CALC:TRAN:TIME:WIND DCH`
Reset the instrument and select a Dolph-Chebyshev window for filtering the data in the frequency domain.
`CALC:TRAN:TIME:DCH 25`
Set the sideband suppression to 25 dB.

Manual operation: See "Side Lobe Level" on page 359

CALCulate<Chn>:TRANsform:TIME:EXPonential <ExponentialTau>

Sets the smoothness factor for the exponential window.

This command is only available if an exponential window is active (`CALCulate<Chn>:TRANsform:TIME:WINDow EXPonential`).

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<ExponentialTau> Decay rate τ
See Table 4-22.

Options: Option R&S ZNB-K2 / R&S ZNBT-K2

Manual operation: See "Exponential Smooth Factor" on page 360

CALCulate<Chn>:TRANsform:TIME:GAUSsian <GaussianSigma>

Sets the smoothness factor for a Gaussian window.

This command is only available if a Gaussian window is active (`CALCulate<Chn>:TRANsform:TIME:WINDow GAUSsian`).

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<GaussianSigma> Standard deviation σ of the original Gaussian function
See Table 4-22.

Options: Option R&S ZNB-K2 / R&S ZNBT-K2

Manual operation: See "Gaussian Smooth Factor" on page 359

CALCulate<Chn>:TRANsform:TIME:KAISer <KaiserBeta>

Sets the smoothness factor for the Kaiser window.

This command is only available if a Kaiser window is active (`CALCulate<Chn>:TRANsform:TIME:WINDow KAISer`).

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<KaiserBeta> Kaiser β value
See Table 4-22.

Options: Option R&S ZNB-K2 / R&S ZNBT-K2

Manual operation: See "Kaiser Smooth Factor" on page 359

CALCulate<Chn>:TRANsform:TIME:LPASs <Algorithm>

Calculates the harmonic grid for low pass time domain transforms according to one of the three alternative algorithms.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<Algorithm> KFSTop | KDFRequency | KSDFrequency

KFSTop - keep stop frequency and number of points
KDFRequency - keep frequency gap and number of points
KSDfrequency - keep stop frequency and approximate frequency gap

Example: See `CALCulate<Chn>:TRANsform:TIME[:TYPE]`

Manual operation: See "Set Harmonic Grid and Keep" on page 361

CALCulate<Chn>:TRANsform:TIME:LPASs:DCSParam <DCValue>

Defines the DC value for low pass transforms. The command is enabled only if the sweep points are on a harmonic grid (to be set explicitly or using `CALCulate<Chn>:TRANsform:TIME:LPASs`).

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<DCValue> DC value of the measured quantity

Range: Depending on the measured quantity (-1 to +1 for S-parameters)
*RST: 0

Example:
```
*RST; :CALC:TRAN:TIME:STAT ON
```
Reset the instrument, activating a frequency sweep with S_{21} as measured quantity, and enable the time domain transformation for the default trace.
```
CALC:TRAN:TIME LPAS; TIME:STIM STEP
```
Select a low pass step transformation.
```
CALC:TRAN:TIME:LPAS KFST
```
Calculate a harmonic grid, maintaining the stop frequency and the number of points.
```
CALC:TRAN:TIME:LPAS:DCSP 0.2
```
Set the DC value.
```
CALC:TRAN:TIME:LPAS:DCSP:EXTR; :CALC:TRAN:TIME:LPAS:DCSP?
```
Extrapolate the measured trace, overwrite the defined DC value, and query the new value.
```
CALC:TRAN:TIME:LPAS:DCSP:CONT ON
```
Switch over to continuous extrapolation (e.g. because you noticed a discrepancy between the manually entered DC value and the extrapolation and assume the extrapolation to be more trustworthy).
```
CALC:TRAN:TIME:RES:EFAC 3
```
Select a resolution enhancement factor of 3 in order to improve the resolution in time domain.

Manual operation: See "DC Value" on page 362

CALCulate<Chn>:TRANsform:TIME:LPASs:DCSParam:CONTinuous <Boolean>

Determines whether continuous extrapolation for the DC value is enabled.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<Boolean> ON - continuous extrapolation enabled
OFF - continuous extrapolation disabled

*RST: ON

Example: See CALCulate<Chn>:TRANsform:TIME:LPASs:DCSParam

Options: R&S ZNB-K2

Manual operation: See "DC Value" on page 362

CALCulate<Chn>:TRANsform:TIME:LPASs:DCSParam:EXTRapolate

Extrapolates the measured trace towards f = 0 and overwrites the current DC value (CALCulate<Chn>:TRANsform:TIME:LPASs:DCSParam). The command is relevant for low pass time domain transforms.

Suffix:
<Chn> Channel number used to identify the active trace

Example: See CALCulate<Chn>:TRANsform:TIME:LPASs:DCSParam

Usage: Event

Manual operation: See "DC Value" on page 362

CALCulate<Chn>:TRANsform:TIME:LPFRequency

Calculates the harmonic grid for low pass time domain transforms, keeping the stop frequency and the number of points.

Tip: Use CALCulate<Chn>:TRANsform:TIME:LPASs if you wish to use one of the other algorithms for calculating the grid.

Suffix:
<Chn> Channel number used to identify the active trace

Example: See CALCulate<Chn>:TRANsform:TIME[:TYPE]

Usage: Event

Manual operation: See "DC Value" on page 362

CALCulate<Chn>:TRANsform:TIME:RESolution:EFACtor <REfactor>

Defines the resolution enhancement factor for the time domain transform.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<REfactor> Resolution enhancement factor.

Range: 1 to 10.
Increment: 0.1
*RST: 1 (no resolution enhancement)

Example: See CALCulate<Chn>:TRANsform:TIME:LPASs:DCSParam

Manual operation: See "Resolution Enh." on page 360

CALCulate<Chn>:TRANsform:TIME:SPAN <Span>

Defines the time span of the diagram in time domain.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<Span> Time span of the diagram in time domain.

Range: 2E-012 s to 200 s.
Increment: 0.1 ns
*RST: 5E-009 s
Default unit: s

Example: See `CALCulate<Chn>:TRANsform:TIME:CENTer`

Manual operation: See "Time Start / Time Stop / Time Center / Time Span" on page 529

Note: If the x-axis is scaled in distance units (`CALCulate<Chn>:TRANsform:TIME:XAXis DISTance`), then the span is entered in m; the range and default value changes accordingly.

CALCulate<Chn>:TRANsform:TIME:STARt <StartTime>

Defines the start time of the diagram in time domain.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<StartTime> Start time of the diagram.

Range: -100 s to +99.999999999998 s.
Increment: 0.1 ns
*RST: -1E-009 s
Default unit: s

Example:
```
*RST; :CALC:TRAN:TIME:STAT ON
```
Reset the instrument, activating a frequency sweep, and enable the time domain transformation for the default trace.
```
CALC:TRAN:TIME:STAR 0; STOP 10 ns
```
Set the start time to 0 ns and the stop time to 10 ns.

Manual operation: See "Time Start / Time Stop / Time Center / Time Span" on page 529

Note: If the x-axis is scaled in distance units (`CALCulate<Chn>:TRANsform:TIME:XAXis DISTance`), then the start value is entered in m; the range and default value changes accordingly.

CALCulate<Chn>:TRANsform:TIME:STATe <Boolean>

Determines whether the time domain transformation for trace no. `<Chn>` is enabled.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<Boolean> ON - time domain representation active.
OFF - frequency domain representation active.
*RST: OFF

Example: `*RST; :CALC:TRAN:TIME:STAT?`
Reset the instrument, activating a frequency sweep, and query whether the default trace is displayed in the time domain. The response is 0.

Manual operation: See "Time Domain" on page 357

CALCulate<Chn>:TRANsform:TIME:STIMulus <Type>

Selects the type of stimulus to be simulated in the low pass transformation process.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<Type> IMPulse | STEP
IMPulse - impulse response, in bandpass or lowpass mode.
STEP - step response, only in lowpass mode (a bandpass mode setting `CALCulate<Chn>:TRANsform:TIME[:TYPE] BPASs` is automatically changed to lowpass).
*RST: IMP

Example: See `CALCulate<Chn>:TRANsform:TIME[:TYPE]`

Manual operation: See "Type" on page 359

CALCulate<Chn>:TRANsform:TIME:STOP <StopTime>

Defines the stop time of the diagram in time domain.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<StopTime> Stop time of the diagram.
Range: -99.999999999998 s to +100 s.
Increment: 0.1 ns
*RST: +4E-009 s
Default unit: s

Example: See `CALCulate<Chn>:TRANsform:TIME:STARt`

Manual operation: See "Time Start / Time Stop / Time Center / Time Span" on page 529

Note: If the x-axis is scaled in distance units (`CALCulate<Chn>:TRANsform:TIME:XAXis DISTance`), then the stop value is entered in m; the range and default value changes accordingly.

CALCulate<Chn>:TRANsform:TIME[:TYPE] <TransformType>

Selects the time domain transformation type.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<TransformType> BPASs | LPASs

BPASs - band pass impulse (only impulse response; a step response `CALCulate<Chn>:TRANsform:TIME:STIMulus STEP` is automatically changed to impulse response)
LPASs - low pass (impulse or step response, depending on `CALCulate<Chn>:TRANsform:TIME:STIMulus` setting)

*RST: BPASs

Example: `*RST; :CALC:TRAN:TIME:STAT ON`
Reset the instrument, activating a frequency sweep, and enable the time domain transformation for the default trace.
`CALC:TRAN:TIME LPAS; TIME:STIM STEP`
Select a low pass step transformation.
`CALC:TRAN:TIME:LPAS KFST`
Calculate a harmonic grid, keeping the stop frequency and the number of points.

Manual operation: See "Type" on page 359

CALCulate<Chn>:TRANsform:TIME:WINDow <WindowType>

Selects the window type for filtering the data in the frequency domain prior to the time domain transformation.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<WindowType> RECT | HAMMing | HANNing | BOHMan | DCHebyshev | CAUSal | BLACkman | KAISer | GAUSsian | EXPonential

See Section 4.7.1.3, "Windows in the frequency domain", on page 220.

RECT
No profiling (rectangle)

HANN
Normal profile (Hann)

HAMMing
Low first sidelobe (Hamming)

BOHMan
Steep falloff (Bohman)

DCHebyshev
Arbitrary sidelobes (Dolph-Chebychev)

CAUSal | BLACkman | KAISer | GAUSsian | EXPonential
Additional window types; see Table 4-22
Note that selecting CAUSal automatically activates low pass step transformation (`CALCulate<Chn>:TRANsform:TIME[:TYPE] LPASs`, `CALCulate<Chn>:TRANsform:TIME:STIMulus ON`) with automatic harmonic grid (`[SENSe<Ch>:]HARMonic:AUTO ON`).

*RST: HANN

Example: See `CALCulate<Chn>:TRANsform:TIME:DCHebyshev`

Manual operation: See "Impulse Response" on page 359

CALCulate<Chn>:TRANsform:TIME:XAXis <Unit>

Switches over between the x-axis scaling in time units or distance units.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<Unit> TIME | DISTance

TIME - x-axis scaled in time units.
DISTance - x-axis scaled in distance units (Distance = Time * c_0 * Velocity Factor).

Example:
```
*RST; :CALC:TRAN:TIME:STAT ON
```
Reset the instrument, activating a frequency sweep, and enable the time domain transformation for the default trace.
```
CALC:TRAN:TIME:XAX DIST
```
Convert the x-axis scaling to distance units.

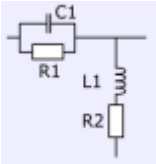
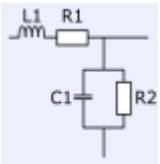
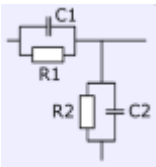
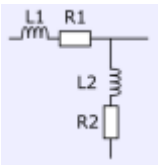
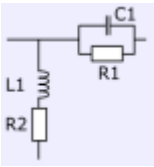
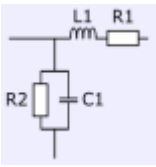
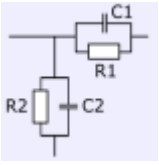
Manual operation: See "Time / Distance" on page 530

CALCulate:TRANsform:VNETworks...

The `CALCulate:TRANsform:VNETworks...` commands define and activate the circuit models for the different de-/embedding types. See Section 4.6.2, "Embedding and deembedding", on page 203.

The circuit models are addressed by predefined enum constants. Except for the FIM-Port model that consists of a single snp file, the models and constants are different for the different de-/embedding types.

Table 7-5: Circuit models for single ended port de-/embedding

Parameter	Circuit model	Pictogram
FIMPort	File import, generic 2-port (no circuit model)	
CSL	Serial C, shunt L	
LSC	Serial L, shunt C	
CSC	Serial C, shunt C	
LSL	Serial L, shunt L	
SLC	Shunt L, serial C	
SCL	Shunt C, serial L	
SCC	Shunt C, serial C	

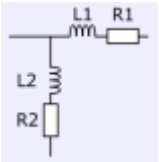
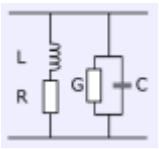
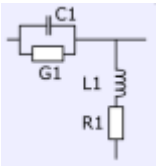
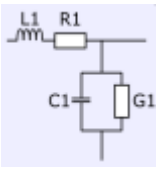
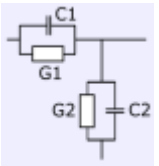
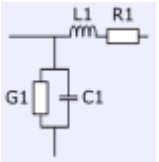
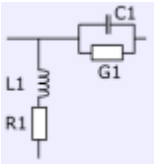
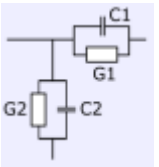
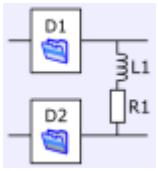
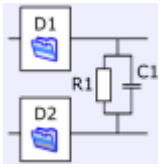
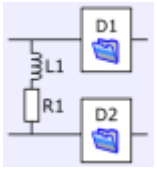
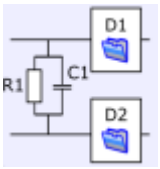
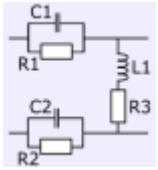
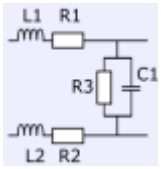
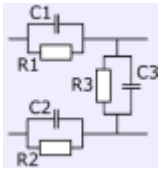
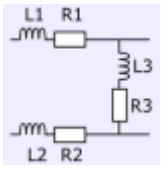
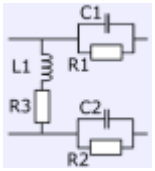
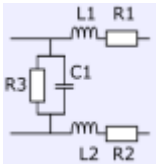
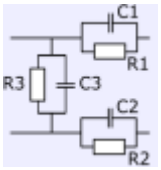
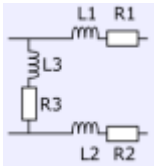
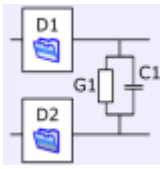
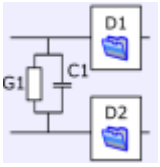
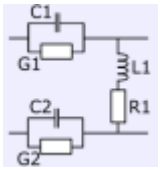
Parameter	Circuit model	Pictogram
SLL	Shunt L, serial L	
SHLC	Shunt L, shunt C	
GSL	Serial C, shunt L	
LSG	Serial L, shunt C	
GSG	Serial C, shunt C	
SGL	Shunt C, serial L	
SLG	Shunt L, serial C	
SGG	Shunt C, serial C	

Table 7-6: Circuit models for balanced port (and port pair) de-/embedding

Parameter	Circuit model	Pictogram
FIMPort	File import, generic 4-port (.s4p, no circuit model)	
STSL	Serial Touchstone (.s2p) data, shunt L	
STSC	Serial Touchstone (.s2p) data, shunt C	
SLST	Shunt L, serial Touchstone (.s2p) data	
SCST	Shunt C, serial Touchstone (.s2p) data	
CSSL	Serial Cs, shunt L	
LSSC	Serial Ls, shunt C	
CSSC	Serial Cs, shunt C	

Parameter	Circuit model	Pictogram
LSSL	Serial Ls, shunt L	
SLCS	Shunt L, serial Cs	
SCLS	Shunt C, serial Ls	
SCCS	Shunt C, serial Cs	
SLLS	Shunt L, serial Ls	
STSG	Serial Touchstone (.s2p) data, shunt C	
SGST	Shunt C, serial Touchstone (.s2p) data	
GSSL	Serial Cs, shunt L	

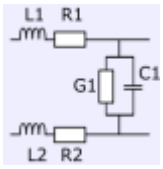
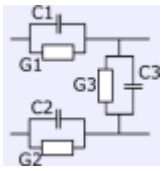
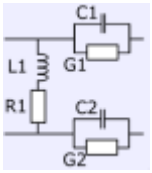
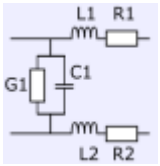
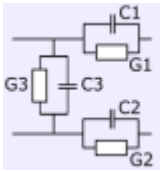
Parameter	Circuit model	Pictogram
LSSG	Serial Ls, shunt C	
GSSG	Serial Cs, shunt C	
SLGS	Shunt L, serial Cs	
SGLS	Shunt C, serial Ls	
SGGS	Shunt C, serial Cs	

Table 7-7: Circuit models for port set de-/embedding

Parameter	Circuit model	Pictogram
FIMPort	File import, no circuit model	
STSL, ..., SGGS	The models for port pairs (i.e. port sets with two ports) are the same as the ones for balanced ports See Table 7-6.	

Table 7-8: Circuit models for ground loop de-/embedding

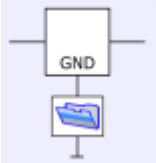
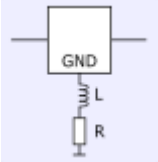
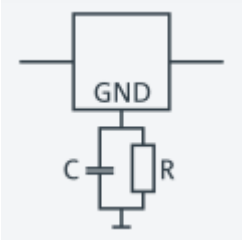
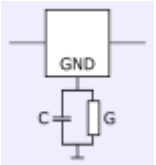
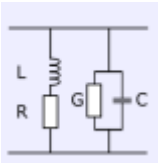
Parameter	Circuit model	Pictogram
FIMPort	File import, no circuit model	
SL	Shunt L	
SC	Shunt C	
SG	Shunt C	

Table 7-9: Circuit models for differential match embedding

Parameter	Circuit model	Pictogram
FIMPort	File import, generic 2-port (no circuit model)	
SHLC	Shunt L, shunt C	

CALCulate<Ch>:TRANsform:VNETworks:ACTivateall[:STATe] 1122
CALCulate<Ch>:TRANsform:VNETworks:BALanced:DEEMbedding<LogPt>:
PARameters:C<Cmp> 1123
CALCulate<Ch>:TRANsform:VNETworks:BALanced:DEEMbedding<LogPt>:
PARameters:DATA<Port> 1124
CALCulate<Ch>:TRANsform:VNETworks:BALanced:DEEMbedding<LogPt>:
PARameters:G<Cmp> 1125

CALCulate<Ch>:TRANsform:VNETworks:BALanced:DEEMbedding<LogPt>:
PARameters:L<Cmp> 1125
CALCulate<Ch>:TRANsform:VNETworks:BALanced:DEEMbedding<LogPt>:
PARameters:R<Cmp> 1126
CALCulate<Ch>:TRANsform:VNETworks:BALanced:DEEMbedding<LogPt>[:STATe] 1127
CALCulate<Ch>:TRANsform:VNETworks:BALanced:DEEMbedding<LogPt>:TNDefinition 1127
CALCulate<Ch>:TRANsform:VNETworks:BALanced:EMBedding<LogPt>:PARameters:
C<Cmp> 1128
CALCulate<Ch>:TRANsform:VNETworks:BALanced:EMBedding<LogPt>:PARameters:
DATA<Port> 1129
CALCulate<Ch>:TRANsform:VNETworks:BALanced:EMBedding<LogPt>:PARameters:
G<Cmp> 1130
CALCulate<Ch>:TRANsform:VNETworks:BALanced:EMBedding<LogPt>:PARameters:
L<Cmp> 1130
CALCulate<Ch>:TRANsform:VNETworks:BALanced:EMBedding<LogPt>:PARameters:
R<Cmp> 1131
CALCulate<Ch>:TRANsform:VNETworks:BALanced:EMBedding<LogPt>[:STATe] 1132
CALCulate<Ch>:TRANsform:VNETworks:BALanced:EMBedding<LogPt>:TNDefinition 1132
CALCulate<Ch>:TRANsform:VNETworks:DIFFerential:EMBedding<LogPt>:
PARameters:C<Cmp> 1133
CALCulate<Ch>:TRANsform:VNETworks:DIFFerential:EMBedding<LogPt>:
PARameters:DATA 1133
CALCulate<Ch>:TRANsform:VNETworks:DIFFerential:EMBedding<LogPt>:
PARameters:G<Cmp> 1134
CALCulate<Ch>:TRANsform:VNETworks:DIFFerential:EMBedding<LogPt>:
PARameters:L<Cmp> 1134
CALCulate<Ch>:TRANsform:VNETworks:DIFFerential:EMBedding<LogPt>:
PARameters:R<Cmp> 1135
CALCulate<Ch>:TRANsform:VNETworks:DIFFerential:EMBedding<LogPt>[:STATe] 1135
CALCulate<Ch>:TRANsform:VNETworks:DIFFerential:EMBedding<LogPt>:TNDefinition 1136
CALCulate<Ch>:TRANsform:VNETworks:FSIMulator[:STATe] 1136
CALCulate<Ch>:TRANsform:VNETworks:GLOop:DEEMbedding<group>:PARameters:C 1136
CALCulate<Ch>:TRANsform:VNETworks:GLOop:DEEMbedding<group>:PARameters:G 1137
CALCulate<Ch>:TRANsform:VNETworks:GLOop:DEEMbedding<group>:PARameters:L 1138
CALCulate<Ch>:TRANsform:VNETworks:GLOop:DEEMbedding<group>:PARameters:R 1138
CALCulate<Ch>:TRANsform:VNETworks:GLOop:DEEMbedding<group>[:STATe] 1139
CALCulate<Ch>:TRANsform:VNETworks:GLOop:DEEMbedding<group>:TNDefinition 1140
CALCulate<Ch>:TRANsform:VNETworks:GLOop:EMBedding<group>:PARameters:C 1140
CALCulate<Ch>:TRANsform:VNETworks:GLOop:EMBedding<group>:PARameters:G 1141
CALCulate<Ch>:TRANsform:VNETworks:GLOop:EMBedding<group>:PARameters:L 1141
CALCulate<Ch>:TRANsform:VNETworks:GLOop:EMBedding<group>:PARameters:R 1142
CALCulate<Ch>:TRANsform:VNETworks:GLOop:EMBedding<group>[:STATe] 1143
CALCulate<Ch>:TRANsform:VNETworks:GLOop:EMBedding<group>:TNDefinition 1143
CALCulate<Ch>:TRANsform:VNETworks:GLOop:GROup 1144
CALCulate<Ch>:TRANsform:VNETworks:PPAir:DEEMbedding<ListId>:DEFine 1144
CALCulate<Ch>:TRANsform:VNETworks:PPAir:DEEMbedding:DELete 1145
CALCulate<Ch>:TRANsform:VNETworks:PPAir:DEEMbedding<ListId>:PARameters:
C<1|2|3> 1145
CALCulate<Ch>:TRANsform:VNETworks:PPAir:DEEMbedding<ListId>:PARameters:
G<1|2|3> 1146

CALCulate<Ch>:TRANsform:VNETworks:PPAir:DEEMbedding<ListId>:PARameters:L<1|2|3> 1147
CALCulate<Ch>:TRANsform:VNETworks:PPAir:DEEMbedding<ListId>:PARameters:R<1|2|3> 1147
CALCulate<Ch>:TRANsform:VNETworks:PPAir:DEEMbedding<ListId>[:STATe] 1148
CALCulate<Ch>:TRANsform:VNETworks:PPAir:DEEMbedding<ListId>:TNDefinition 1149
CALCulate<Ch>:TRANsform:VNETworks:PPAir:EMBedding<ListId>:DEFine 1149
CALCulate<Ch>:TRANsform:VNETworks:PPAir:EMBedding<ListId>:DELete 1150
CALCulate<Ch>:TRANsform:VNETworks:PPAir:EMBedding<ListId>:PARameters:C<1|2|3> 1150
CALCulate<Ch>:TRANsform:VNETworks:PPAir:EMBedding<ListId>:PARameters:L<1|2|3> 1151
CALCulate<Ch>:TRANsform:VNETworks:PPAir:EMBedding<ListId>:PARameters:R<1|2|3> 1151
CALCulate<Ch>:TRANsform:VNETworks:PPAir:EMBedding<ListId>:PARameters:G<1|2|3> 1152
CALCulate<Ch>:TRANsform:VNETworks:PPAir:EMBedding<ListId>[:STATe] 1153
CALCulate<Ch>:TRANsform:VNETworks:PPAir:EMBedding<ListId>:TNDefinition 1153
CALCulate<Ch>:TRANsform:VNETworks:PSET:DEEMbedding<ListId>:DEFine 1154
CALCulate<Ch>:TRANsform:VNETworks:PSET:EMBedding<ListId>:DEFine 1154
CALCulate<Ch>:TRANsform:VNETworks:SENDed:DEEMbedding<PhyPt>:IGNore 1154
CALCulate<Ch>:TRANsform:VNETworks:SENDed:DEEMbedding<PhyPt>:PARameters:C<Cmp> 1155
CALCulate<Ch>:TRANsform:VNETworks:SENDed:DEEMbedding<PhyPt>:PARameters:DATA 1155
CALCulate<Ch>:TRANsform:VNETworks:SENDed:DEEMbedding<PhyPt>:PARameters:G<Cmp> 1156
CALCulate<Ch>:TRANsform:VNETworks:SENDed:DEEMbedding<PhyPt>:PARameters:L<Cmp> 1157
CALCulate<Ch>:TRANsform:VNETworks:SENDed:DEEMbedding<PhyPt>:PARameters:R<Cmp> 1158
CALCulate<Ch>:TRANsform:VNETworks:SENDed:DEEMbedding<PhyPt>[:STATe] 1158
CALCulate<Ch>:TRANsform:VNETworks:SENDed:DEEMbedding<PhyPt>:TNDefinition 1159
CALCulate<Ch>:TRANsform:VNETworks:SENDed:EMBedding<PhyPt>:PARameters:C<Cmp> 1159
CALCulate<Ch>:TRANsform:VNETworks:SENDed:EMBedding<PhyPt>:PARameters:DATA 1160
CALCulate<Ch>:TRANsform:VNETworks:SENDed:EMBedding<PhyPt>:PARameters:G<Cmp> 1161
CALCulate<Ch>:TRANsform:VNETworks:SENDed:EMBedding<PhyPt>:PARameters:L<Cmp> 1161
CALCulate<Ch>:TRANsform:VNETworks:SENDed:EMBedding<PhyPt>:PARameters:R<Cmp> 1162
CALCulate<Ch>:TRANsform:VNETworks:SENDed:EMBedding<PhyPt>[:STATe] 1163
CALCulate<Ch>:TRANsform:VNETworks:SENDed:EMBedding<PhyPt>:TNDefinition 1163
CALCulate<Ch>:TRANsform:VNETworks:WAVes 1164

CALCulate<Ch>:TRANsform:VNETworks:ACTivateall[:STATe] <Enable>

Deactivates or activates all deembeddings (single-ended, balanced, port set, ground loop) configured for channel `<Ch>`.

Suffix:
<Ch> Channel number

Parameters:
<Enable> **ON | 1**
Activate all deembeddings
OFF | 0
Deactivate all deembeddings

Manual operation: See "All Deembedding Activated" on page 704

CALCulate<Ch>:TRANsform:VNETworks:BALanced:DEEMbedding<LogPt>:PARameters:C<Cmp> <CircuitModel>, <Capacitance>
CALCulate<Ch>:TRANsform:VNETworks:BALanced:DEEMbedding<LogPt>:PARameters:C<Cmp>? <CircuitModel>

Specifies the capacitance value C<Cmp> in the different circuit models for balanced port deembedding.

In the query form, the <Capacitance> parameter must be omitted. The command returns the capacitance value for the specified circuit model.

Suffix:
<Ch> Channel number

<LogPt> Logical port number (balanced port)

<Cmp> Number of capacitance in circuit model. The total number of capacitances depends on the selected circuit model.

Parameters:
<Capacitance> Capacitance C<Cmp> for the specified circuit model.
Range: -1mF to 1 mF.
Increment: 1 fF (1E-15 F)
*RST: 1 pF (1E-12 F)
Default unit: F

Parameters for setting and query:
<CircuitModel> STSC | SCST | CSSL | LSSC | CSSC | SLCS | SCLS | SCCS | STSG | SGST | GSSL | LSSG | GSSG | SLGS | SGLS | SGGS
Possible circuit models (character data); see Table 7-6.

Example:
```
*RST; :SOUR:LPOR1 1,2; :CALC:TRAN:VNET:BAL:DEEM:PAR:C2? CSSL
```
Create a balanced port and query the default capacitance C2 for the Serial Cs, shunt L circuit model. The response is 1E-012 (1 pF).
```
CALC:TRAN:VNET:BAL:DEEM:PAR:C2 CSSL, 2.2E-12
```
Increase the capacitance to 2.2 pF.

Manual operation: See "Network" on page 753

CALCulate<Ch>:TRANsform:VNETworks:BALanced:DEEMbedding<LogPt>:PARameters:DATA<Port> <Interchange>, <arg1>

Defines a deembedding network for a balanced port based on the given S-Parameter traces.

Circuit models `STSL | STSC | SLST | SCST` require S-Parameter traces of two 2-port networks, to be assigned to the different ports `PMAin` and `PSECondary`; the `FIMPort` model requires S-Parameter traces of a single 4-port network but no additional port assignment.

Use

- `CALCulate<Ch>:TRANsform:VNETworks:BALanced:DEEMbedding<LogPt>:TNDefinition` to select the adequate circuit model **before** executing this command.
- `MMEMory:LOAD:VNETworks<Ch>:BALanced:DEEMbedding<LogPt>` to load circuit data from a Touchstone file located at the R&S ZNB's file system.

Suffix:

<Ch> Channel number

<LogPt> Logical port number

<Port> Port assignment for two 2-port networks:
1 - Port 1
2 - Port 2
This suffix is ignored for 4-port networks.

Setting parameters:

<Interchange> FPORts | IPORts | SGATes | SINCreasing

FPORts (or omitted)
Standard port sequence (odd port numbers towards VNA, even port numbers towards DUT)

IPORts
– Two-port networks: inverted port sequence (network port 2 towards VNA, network port 1 towards DUT)
– Four-port networks: increasing port sequence (low port numbers towards VNA, high port numbers towards DUT)

SGATes
Swapped gates (even port numbers towards VNA, odd port numbers towards DUT)

SINCreasing
Swapped increasing port sequence (high port numbers towards VNA, low port numbers towards DUT)

<arg1> <block_data>

Content of a Touchstone file (`*.s2p` or `*.s4p`) in IEEE488.2 Block data format.

Usage: Setting only

CALCulate<Ch>:TRANsform:VNETworks:BALanced:DEEMbedding<LogPt>:PARameters:G<Cmp> <CircuitModel>, <Conductance>
CALCulate<Ch>:TRANsform:VNETworks:BALanced:DEEMbedding<LogPt>:PARameters:G<Cmp>? <CircuitModel>

Specifies the conductance value G<Cmp> in the different circuit models for balanced port deembedding.

In the query form, the <Conductance> parameter must be omitted. The command returns the capacitance value for the specified circuit model.

Suffix:

<Ch> Channel number

<LogPt> Logical port number (balanced port)

<Cmp> Number of conductance in circuit model. The total number of conductances depends on the selected circuit model.

Parameters:

<Conductance> Conductance G<Cmp> for the specified circuit model.

Range: -1kS to 1 kS.
Increment: 1 nS (1E-9 S)
*RST: 0 S
Default unit: Siemens (SI unit symbol: S)

Parameters for setting and query:

<CircuitModel> STSG | SGST | GSSL | LSSG | GSSG | SLGS | SGLS | SGGS

Possible circuit models (character data); see Table 7-6.

Manual operation: See "Network" on page 753

CALCulate<Ch>:TRANsform:VNETworks:BALanced:DEEMbedding<LogPt>:PARameters:L<Cmp> <CircuitModel>, <Inductance>
CALCulate<Ch>:TRANsform:VNETworks:BALanced:DEEMbedding<LogPt>:PARameters:L<Cmp>? <CircuitModel>

Specifies the inductance value L<Cmp> in the different circuit models for balanced port deembedding.

In the query form, the <Inductance> parameter must be omitted. The command returns the inductance value for the specified circuit model.

Suffix:

<Ch> Channel number

<LogPt> Logical port number (balanced port)

<Cmp> Number of inductance in circuit model. The total number of inductances depends on the selected circuit model.

Parameters:

<Inductance> Inductance L<Cmp> for the specified circuit model.

Range: -1H to 1 H.
Increment: 1 pH (1E-12 H)
*RST: 1 nH (1E-9 H)
Default unit: H

Parameters for setting and query:

<CircuitModel> STSL | SLST | CSSL | LSSC | LSSL | SLCS | SCLS | SLLS | GSSL | LSSG | SLGS | SGLS

Possible circuit models (character data); see Table 7-6.

Example:

```
*RST; :SOUR:LPOR1 1,2; :CALC:TRAN:VNET:BAL:DEEM:PAR:L1? CSSL
```

Create a balanced port and query the default inductance L1 for the Serial Cs, shunt L circuit model. The response is 1E-009 (1 nH).

```
CALC:TRAN:VNET:BAL:DEEM:PAR:L1 CSSL, 2.2E-9
```

Increase the inductance to 2.2 nH.

Manual operation: See "Network" on page 753

CALCulate<Ch>:TRANsform:VNETworks:BALanced:DEEMbedding<LogPt>:PARameters:R<Cmp> <CiruitModel>, <Resistance>
CALCulate<Ch>:TRANsform:VNETworks:BALanced:DEEMbedding<LogPt>:PARameters:R<Cmp>? <CiruitModel>

Specifies the resistance value R<Cmp> in the different circuit models for balanced port deembedding.

In the query form, the <Resistance> parameter must be omitted. The command returns the resistance value for the specified circuit model.

Suffix:

<Ch> Channel number

<LogPt> Logical port number (balanced port)

<Cmp> Number of resistance in circuit model. The total number of resistances depends on the selected circuit model.

Parameters:

<Resistance> Resistance R<Cmp> for the specified circuit model.

Range: -10 MΩ to 10 MΩ.
Increment: 1 mΩ (1E-3 Ω)
*RST: 0 Ω for all resistances connected in series with an inductance. 10 MΩ for all resistances connected in parallel with a capacitance
Default unit: Ω

Parameters for setting and query:

<CiruitModel> STSL | STSC | SLST | SCST | CSSL | LSSC | CSSC | LSSL | SLCS | SCLS | SCCS | SLLS | GSSL | LSSG | SLGS | SGLS

Possible circuit models (character data); see Table 7-6.

Example: `*RST; :SOUR:LPOR1 1,2; :CALC:TRAN:VNET:BAL:DEEM:PAR:R1? CSSL; R2? CSSL; R3? CSSL`
Create a balanced port and query the default resistances for the Serial Cs, shunt L circuit model. The response is 10000000000;10000000000; 0.
`CALC:TRAN:VNET:BAL:DEEM:PAR:R3 CSSL, 2.2E+3`
Increase the resistance R3 to 2.2 kΩ.

Manual operation: See "Network" on page 753

CALCulate<Ch>:TRANsform:VNETworks:BALanced:DEEMbedding<LogPt>[:STATe] <Boolean>

Enables or disables the deembedding function for balanced ports. It is allowed to change the circuit model and its parameters while deembedding is enabled.

Suffix:

<Ch> Channel number

<LogPt> Logical port number (balanced port)

Parameters:

<Boolean> ON - deembedding active
OFF - deembedding inactive
*RST: OFF

Example: `*RST; SOUR:LPOR1 1,2; LPOR2 3,4`
Define a balanced port configuration.
`CALC:TRAN:VNET:BAL:DEEM:TND CSSL`
Select the Serial Cs, shunt L circuit model for deembedding.
`CALC:TRAN:VNET:BAL:DEEM:PAR:R3 CSSL, 2.2E+3; :CALC:TRAN:VNET:BAL:DEEM ON`
Increase the resistance R3 for the Serial Cs, shunt L circuit model to 2.2 kΩ and enable deembedding.

Manual operation: See "Active" on page 711

CALCulate<Ch>:TRANsform:VNETworks:BALanced:DEEMbedding<LogPt>:TNDefinition <CircuitModel>

Selects the circuit model for balanced port deembedding.

Suffix:

<Ch> Channel number

<LogPt> Logical port number (balanced port)

Parameters:

<CircuitModel> FIMPort | STSL | STSC | SLST | SCST | CSSL | LSSC | CSSC | LSSL | SLCS | SCLS | SCCS | SLLS | STSG | SGST | GSSL | LSSG | GSSG | SLGS | SGLS | SGGS

Possible circuit models (character data); see Table 7-6.

*RST: CSSL

Example: See `CALCulate<Ch>:TRANsform:VNETworks:BALanced:DEEMbedding<LogPt>[:STATe]`

Manual operation: See "Network" on page 753

CALCulate<Ch>:TRANsform:VNETworks:BALanced:EMBedding<LogPt>:PARameters:C<Cmp> <CiruitModel>, <Capacitance>
CALCulate<Ch>:TRANsform:VNETworks:BALanced:EMBedding<LogPt>:PARameters:C<Cmp>? <CiruitModel>

Specifies the capacitance value C<Cmp> in the different circuit models for balanced port embedding.

In the query form, the <Capacitance> parameter must be omitted. The command returns the capacitance value for the specified circuit model.

Suffix:

<Ch> Channel number

<LogPt> Logical port number (balanced port)

<Cmp> Number of capacitance in circuit model. The total number of capacitances depends on the selected circuit model.

Parameters:

<Capacitance> Capacitance C<Cmp> for the specified circuit model.

Range: -1mF to 1 mF.
Increment: 1 fF (1E-15 F)
*RST: 1 pF (1E-12 F)
Default unit: F

Parameters for setting and query:

<CiruitModel> STSC | SCST | CSSL | LSSC | CSSC | SLCS | SCLS | SCCS | STSG | SGST | GSSL | LSSG | GSSG | SLGS | SGLS | SGGS

Possible circuit models (character data); see Table 7-6.

Example: `*RST; :SOUR:LPOR1 1,2; :CALC:TRAN:VNET:BAL:EMB:PAR:C2? CSSL`
Create a balanced port and query the default capacitance C2 for the Serial Cs, shunt L circuit model. The response is 1E-012 (1 pF).
`CALC:TRAN:VNET:BAL:EMB:PAR:C2 CSSL, 2.2E-12`
Increase the capacitance to 2.2 pF.

Manual operation: See "Network" on page 753

CALCulate<Ch>:TRANsform:VNETworks:BALanced:EMBedding<LogPt>:PARameters:DATA<Port> <Interchange>, <arg1>

Defines an embedding network for a balanced port based on the given S-Parameter traces.

Circuit models `STSL | STSC | SLST | SCST` require S-Parameter traces of two 2-port networks, to be assigned to the different ports; the `FIMPort` model requires S-Parameter traces of a single 4-port network but no additional port assignment.

Use

- `CALCulate<Ch>:TRANsform:VNETworks:BALanced:EMBedding<LogPt>:TNDefinition` to select the adequate circuit model **before** executing this command.
- `MMEMory:LOAD:VNETworks<Ch>:BALanced:EMBedding<LogPt>` to load circuit data from a Touchstone file located at the R&S ZNB's file system.

Suffix:

<Ch> Channel number

<LogPt> Logical port number

<Port> Port assignment for two 2-port networks:
1 - Port 1
2 - Port 2
This suffix is ignored for 4-port networks.

Setting parameters:

<Interchange> FPORts | IPORts | SGATes | SINCreasing

FPORts (or omitted)
Standard port sequence (odd port numbers towards VNA, even port numbers towards DUT)

IPORts
– Two-port networks: inverted port sequence (network port 2 towards VNA, network port 1 towards DUT)
– Four-port networks: increasing port sequence (low port numbers towards VNA, high port numbers towards DUT)

SGATes
Swapped gates (even port numbers towards VNA, odd port numbers towards DUT)

SINCreasing
Swapped increasing port sequence (high port numbers towards VNA, low port numbers towards DUT)

<arg1> <block_data>

Content of a Touchstone file (`*.s2p` or `*.s4p`) in IEEE488.2 Block data format.

Usage: Setting only

CALCulate<Ch>:TRANsform:VNETworks:BALanced:EMBedding<LogPt>:PARameters:G<Cmp> <CircuitModel>, <Conductance>
CALCulate<Ch>:TRANsform:VNETworks:BALanced:EMBedding<LogPt>:PARameters:G<Cmp>? <CircuitModel>

Specifies the conductance value G<Cmp> in the different circuit models for balanced port embedding.

In the query form, the <Conductance> parameter must be omitted. The command returns the capacitance value for the specified circuit model.

Suffix:
<Ch> Channel number

<LogPt> Logical port number (balanced port)

<Cmp> Number of conductance in circuit model. The total number of conductances depends on the selected circuit model.

Parameters:
<Conductance> Conductance G<Cmp> for the specified circuit model.

Range: -1kS to 1 kS.
Increment: 1 pS (1E-12 S)
*RST: 0 S
Default unit: Siemens (SI unit symbol: S)

Parameters for setting and query:
<CircuitModel> STSG | SGST | GSSL | LSSG | GSSG | SLGS | SGLS | SGGS

Possible circuit models (character data); see Table 7-6.

Manual operation: See "Network" on page 753

CALCulate<Ch>:TRANsform:VNETworks:BALanced:EMBedding<LogPt>:PARameters:L<Cmp> <CircuitModel>, <Inductance>
CALCulate<Ch>:TRANsform:VNETworks:BALanced:EMBedding<LogPt>:PARameters:L<Cmp>? <CircuitModel>

Specifies the inductance values L1, L2, L3 in the different circuit models for balanced port embedding.

In the query form, the <Inductance> parameter must be omitted. The command returns the inductance value for the specified circuit model.

Suffix:
<Ch> Channel number

<LogPt> Logical port number (balanced port)

<Cmp> Number of inductance in circuit model. The total number of inductances depends on the selected circuit model.

Parameters:

<Inductance> Inductance L<Cmp> for the specified circuit model.

Range: -1H to 1 H.
Increment: 1 pH (1E-12 H)
*RST: 1 nH (1E-9 H)
Default unit: H

Parameters for setting and query:

<CircuitModel> STSL | SLST | CSSL | LSSC | LSSL | SLCS | SCLS | SLLS | GSSL | LSSG | SLGS | SGLS

Possible circuit models (character data); see Table 7-6.

Example:

```
*RST; :SOUR:LPOR1 1,2; :CALC:TRAN:VNET:BAL:EMB:PAR:L1? CSSL
```

Create a balanced port and query the default inductance L1 for the Serial Cs, shunt L circuit model. The response is 1E-009 (1 nH).

```
CALC:TRAN:VNET:BAL:EMB:PAR:L1 CSSL, 2.2E-9
```

Increase the inductance to 2.2 nH.

Manual operation: See "Network" on page 753

CALCulate<Ch>:TRANsform:VNETworks:BALanced:EMBedding<LogPt>:PARameters:R<Cmp> <CircuitModel>, <Resistance>
CALCulate<Ch>:TRANsform:VNETworks:BALanced:EMBedding<LogPt>:PARameters:R<Cmp>? <CircuitModel>

Specifies the resistance values R1, R2, R3 in the different circuit models for balanced port embedding.

In the query form, the <Resistance> parameter must be omitted. The command returns the resistance value for the specified circuit model.

Suffix:

<Ch> Channel number

<LogPt> Logical port number (balanced port)

<Cmp> Number of resistance in circuit model. The total number of resistances depends on the selected circuit model.

Parameters:

<Resistance> Resistance R<Cmp> for the specified circuit model.

Range: -10 MΩ to 10 MΩ.
Increment: 1 mΩ (1E-3 Ω)
*RST: 0 Ω for all resistances connected in series with an inductance. 10 MΩ for all resistances connected in parallel with a capacitance
Default unit: Ω

Parameters for setting and query:

<CircuitModel> STSL | STSC | SLST | SCST | CSSL | LSSC | CSSC | LSSL | SLCS | SCLS | SCCS | SLLS | GSSL | LSSG | SLGS | SGLS

Possible circuit models (character data); see Table 7-6.

Example:

```
*RST; :SOUR:LPOR1 1,2; :CALC:TRAN:VNET:BAL:EMB:PAR:R1? CSSL; R2? CSSL; R3? CSSL
```

Create a balanced port and query the default resistances for the Serial Cs, shunt L circuit model. The response is 10000000000;10000000000; 0.

```
CALC:TRAN:VNET:BAL:EMB:PAR:R3 CSSL, 2.2E+3
```

Increase the resistance R3 to 2.2 kΩ.

Manual operation: See "Network" on page 753

CALCulate<Ch>:TRANsform:VNETworks:BALanced:EMBedding<LogPt>[:STATe] <Boolean>

Enables or disables the embedding function for balanced ports. It is allowed to change the circuit model and its parameters while embedding is enabled.

Suffix:

<Ch> Channel number

<LogPt> Logical port number (balanced port)

Parameters:

<Boolean> ON | OFF - embedding active or inactive

*RST: OFF

Example:

```
*RST; SOUR:LPOR1 1,2; LPOR2 3,4
```

Define a balanced port configuration.

```
CALC:TRAN:VNET:BAL:EMB:TND CSSL
```

Select the Serial Cs, shunt L circuit model for embedding.

```
CALC:TRAN:VNET:BAL:EMB:PAR:R3 CSSL, 2.2E+3; :CALC:TRAN:VNET:BAL:EMB ON
```

Increase the resistance R3 for the Serial Cs, shunt L circuit model to 2.2 kΩ and enable embedding.

Manual operation: See "Active" on page 711

CALCulate<Ch>:TRANsform:VNETworks:BALanced:EMBedding<LogPt>:TNDefinition <CircuitModel>

Selects the circuit model for balanced port embedding.

Suffix:

<Ch> Channel number

<LogPt> Logical port number (balanced port)

Parameters:

<CircuitModel> FIMPort | STSL | STSC | SLST | SCST | CSSL | LSSC | CSSC | LSSL | SLCS | SCLS | SCCS | SLLS | STSG | SGST | GSSL | LSSG | GSSG | SLGS | SGLS | SGGS

Possible circuit models (character data); see Table 7-6.

*RST: CSSL

Example: See `CALCulate<Ch>:TRANsform:VNETworks:BALanced:EMBedding<LogPt>[:STATe]`

Manual operation: See "Network" on page 753

CALCulate<Ch>:TRANsform:VNETworks:DIFFerential:EMBedding<LogPt>:PARameters:C<Cmp> <CircuitModel>[, <Capacitance>]

Specifies the capacitance value C in the "Shunt L, Shunt C" lumped element model for differential match embedding.

In the query form, the <Capacitance> parameter must be omitted.

Suffix:

<Ch> Channel number

<LogPt> Logical port number (balanced port)

<Cmp> Must be omitted or set to 1.

Parameters:

<CircuitModel> SHLC

Currently only the "Shunt L, Shunt C" lumped element model is supported

<Capacitance> Range: -1mF to 1 mF.
Increment: 1 fF (1E-15 F)
*RST: 1 pF (1E-12 F)
Default unit: F

Manual operation: See "Network" on page 758

CALCulate<Ch>:TRANsform:VNETworks:DIFFerential:EMBedding<LogPt>:PARameters:DATA <Interchange>, <SParamTrcs>

Defines a Differential match embedding network for a balanced port based on the given S-Parameter traces.

Use `MMEMory:LOAD:VNETworks<Ch>:DIFFerential:EMBedding<LogPt>` to load circuit data from a Touchstone file located at the R&S ZNB's file system instead.

Suffix:

<Ch> Channel number

<LogPt> Logical port number of a balanced port

Setting parameters:

<Interchange> FPORts | IPORts | SGATes

FPORts (or omitted)
Standard port sequence (network port 1 towards VNA, network port 2 towards DUT)

IPORts | SGATes
Inverted port sequence (network port 2 towards VNA, network port 1 towards DUT)

<SParamTrcs> <block_data>

Content of a two-port Touchstone file (`*.s2p`) in IEEE488.2 Block data format.

Usage: Setting only

CALCulate<Ch>:TRANsform:VNETworks:DIFFerential:EMBedding<LogPt>:PARameters:G<Cmp> <CircuitModel>[, <Conductance>]

Specifies the conductance value G in the "Shunt L, Shunt C" lumped element model for differential match embedding.

In the query form, the <Conductance> parameter must be omitted.

Suffix:

<Ch> Channel number

<LogPt> Logical port number (balanced port)

<Cmp> Must be omitted or set to 1.

Parameters:

<CircuitModel> SHLC

Currently only the "Shunt L, Shunt C" lumped element model is supported

<Conductance> Range: -10 MS to 10 MS.
Increment: 1 µS (1E-6 F)
*RST: 0 S
Default unit: S(iemens)

Manual operation: See "Network" on page 758

CALCulate<Ch>:TRANsform:VNETworks:DIFFerential:EMBedding<LogPt>:PARameters:L<Cmp> <CircuitModel>[, <Inductance>]

Specifies the inductance value L in the "Shunt L, Shunt C" lumped element model for differential match embedding.

In the query form, the <Inductance> parameter must be omitted.

Suffix:

<Ch> Channel number

<LogPt> Logical port number (balanced port)

<Cmp> Must be omitted or set to 1

Parameters:

<CircuitModel> SHLC

Currently only the "Shunt L, Shunt C" lumped element model is supported

<Inductance> Range: -1 H to 1 H
Increment: 1 pH (1E-12 H)
*RST: 1 nH (1E-9 H)
Default unit: H

Manual operation: See "Network" on page 758

CALCulate<Ch>:TRANsform:VNETworks:DIFFerential:EMBedding<LogPt>:PARameters:R<Cmp> <CircuitModel>[, <Resistance>]

Specifies the resistance value R in the "Shunt L, Shunt C" lumped element model for differential match embedding.

In the query form, the <Resistance> parameter must be omitted.

Suffix:

<Ch> Channel number

<LogPt> Logical port number (balanced port)

<Cmp> Must be omitted or set to 1.

Parameters:

<CircuitModel> SHLC

Currently only the "Shunt L, Shunt C" lumped element model is supported

<Resistance> Range: -10 MΩ to 10 MΩ.
Increment: 1 mΩ (1E-3 Ω)
*RST: 0 Ω
Default unit: Ohm

Manual operation: See "Network" on page 758

CALCulate<Ch>:TRANsform:VNETworks:DIFFerential:EMBedding<LogPt>[:STATe] <Boolean>

Enables or disables differential match embedding for balanced port <LogPt>.

Suffix:

<Ch> Channel number

<LogPt> Logical port number (balanced port)

Parameters:
<Boolean> ON | OFF - embedding active or inactive
*RST: OFF

Manual operation: See "Active" on page 714

CALCulate<Ch>:TRANsform:VNETworks:DIFFerential:EMBedding<LogPt>:TNDefinition <CircuitModel>

Selects the circuit model for differential match embedding.

Suffix:
<Ch> Channel number
<LogPt> Logical port number (balanced port)

Parameters:
<CircuitModel> FIMPort | SHLC
Possible circuit models (character data), see Circuit models for differential match embedding

Manual operation: See "Network" on page 758

CALCulate<Ch>:TRANsform:VNETworks:FSIMulator[:STATe] <Enable>

De/activates the "Fixture Simulator" switch that allows to disable and (re-)enable the configured deembedding, embedding, balanced ports, and port impedance settings for the selected channel.

Suffix:
<Ch> Channel number

Parameters:
<Enable>

Manual operation: See "Fixture Simulator" on page 672

CALCulate<Ch>:TRANsform:VNETworks:GLOop:DEEMbedding<group>:PARameters:C <CircuitModel>, <Capacitance>
CALCulate<Ch>:TRANsform:VNETworks:GLOop:DEEMbedding<group>:PARameters:C? <CircuitModel>

Specifies the capacitance value C in the different circuit models for ground loop embedding.

Suffix:
<Ch> Channel number
<group> Port group (DUT) number.
If multiple port groups are configured (see `SOURce<Ch>:GROup<Grp>:PPORts`) and `CALCulate<Ch>:TRANsform:VNETworks:GLOop:GROup` is `ON`, then each port group can have its own de-/embedding models.

Parameters:

<Capacitance> Capacitance C for ground loop deembedding.

Range: -1mF to 1 mF.
Increment: 1 fF (1E-15 F)
*RST: 1 pF (1E-12 F)
Default unit: F

Parameters for setting and query:

<CircuitModel> SC | SG

Possible circuit models (character data); see Table 7-8.

Example: `*RST; :CALC:TRAN:VNET:GLO:DEEM:PAR:C? SC`
Query the default capacitance for ground loop deembedding.
The response is 1E-012 (1 pF).
`CALC:TRAN:VNET:GLO:DEEM:PAR:C SC, 2.2E-12`
Increase the capacitance to 2.2 pF.

Manual operation: See "Network" on page 755

CALCulate<Ch>:TRANsform:VNETworks:GLOop:DEEMbedding<group>:PARameters:G <CircuitModel>, <Conductance>
CALCulate<Ch>:TRANsform:VNETworks:GLOop:DEEMbedding<group>:PARameters:G? <CircuitModel>

Specifies the conductance value G in the different circuit models for ground loop embedding.

Suffix:

<Ch> Channel number.

<group> Port group (DUT) number.
If multiple port groups are configured (see `SOURce<Ch>:GROup<Grp>:PPORts`) and `CALCulate<Ch>:TRANsform:VNETworks:GLOop:GROup` is `ON`, then each port group can have its own de-/embedding models.

Parameters:

<Conductance> Conductance G for the specified circuit model.

Range: -1kS to 1 kS.
Increment: 1 nS (1E-9 S)
*RST: 0 S
Default unit: Siemens (SI unit symbol: S)

Parameters for setting and query:

<CircuitModel> SG

Possible circuit models (character data); see Table 7-8.

Manual operation: See "Network" on page 755

CALCulate<Ch>:TRANsform:VNETworks:GLOop:DEEMbedding<group>:
PARameters:L <CircuitModel>, <Inductance>
CALCulate<Ch>:TRANsform:VNETworks:GLOop:DEEMbedding<group>:
PARameters:L? <CircuitModel>

Specifies the inductance value in the different circuit models for ground loop deembedding.

Suffix:
<Ch> Channel number.

<group> Port group (DUT) number.
If multiple port groups are configured (see `SOURce<Ch>:GROup<Grp>:PPORts`) and `CALCulate<Ch>:TRANsform:VNETworks:GLOop:GROup` is `ON`, then each port group can have its own de-/embedding models.

Parameters:
<Inductance> Inductance L for ground loop deembedding.

Range: -1 H to 1 H.
Increment: 1 pH (1E-12 H)
*RST: 1 nH (1E-9 H)
Default unit: H

Parameters for setting and query:
<CircuitModel> SL

Possible circuit models (character data); see Table 7-8.

Example:
```
*RST; :CALC:TRAN:VNET:GLO:DEEM:PAR:L? SL
```
Query the default inductance for ground loop deembedding. The response is 1E-009 (1 nH).
```
CALC:TRAN:VNET:GLO:DEEM:PAR:L SL, 2.2E-9
```
Increase the inductance to 2.2 nH.

Manual operation: See "Network" on page 755

CALCulate<Ch>:TRANsform:VNETworks:GLOop:DEEMbedding<group>:
PARameters:R <CircuitModel>, <Resistance>
CALCulate<Ch>:TRANsform:VNETworks:GLOop:DEEMbedding<group>:
PARameters:R? <CircuitModel>

Specifies the resistance value R in the different circuit models for ground loop deembedding.

Suffix:
<Ch> Channel number.

<group> Port group (DUT) number.
If multiple port groups are configured (see `SOURce<Ch>:GROup<Grp>:PPORts`) and `CALCulate<Ch>:TRANsform:VNETworks:GLOop:GROup` is `ON`, then each port group can have its own de-/embedding models.

Parameters:

<Resistance> Resistance R for the specified circuit model.

Range: -10 MΩ to 10 MΩ
Increment: 1 mΩ
*RST: 0 Ω if the resistance is connected in series with an inductance. 10 MΩ if the resistance is connected in parallel with a capacitance)
Default unit: Ω

Parameters for setting and query:

<CircuitModel> SL | SC

Possible circuit models (character data); see Table 7-8.

Example:

```
*RST; :CALC:TRAN:VNET:GLO:DEEM:PAR:R? SC; R? SL
```

Query the default resistances for ground loop deembedding. The response is 10000000; 0.

```
CALC:TRAN:VNET:GLO:DEEM:PAR:R SC, 2.2E+3
```

Increase the resistance for the Shunt C model to 2.2 kΩ.

Manual operation: See "Network" on page 755

CALCulate<Ch>:TRANsform:VNETworks:GLOop:DEEMbedding<group>[:STATe] <Boolean>

Enables or disables the deembedding function for ground loops. It is allowed to change the circuit model and its parameters while deembedding is enabled.

Suffix:

<Ch> Channel number.

<group> Port group (DUT) number.
If multiple port groups are configured (see `SOURce<Ch>:GROup<Grp>:PPORts`) and `CALCulate<Ch>:TRANsform:VNETworks:GLOop:GROup` is `ON`, then each port group can have its own de-/embedding models.

Parameters:

<Boolean> ON - Deembedding active
OFF - Deembedding inactive

*RST: OFF

Example:

```
CALC:TRAN:VNET:GLO:DEEM:TND SL
```

Select the Shunt L circuit model for deembedding.

```
CALC:TRAN:VNET:GLO:DEEM:PAR:R SL, 2.2E+3; :
CALC:TRAN:VNET:GLO:DEEM ON
```

Increase the resistance for the Shunt L circuit model to 2.2 kΩ and enable deembedding.

Manual operation: See "Active" on page 712

CALCulate<Ch>:TRANsform:VNETworks:GLOop:DEEMbedding<group>:TNDefinition <CircuitModel>

Selects the circuit model for ground loop deembedding.

Suffix:

<Ch> Channel number.

<group> Port group (DUT) number.
If multiple port groups are configured (see `SOURce<Ch>:GROup<Grp>:PPORts`) and `CALCulate<Ch>:TRANsform:VNETworks:GLOop:GROup` is `ON`, then each port group can have its own de-/embedding models.

Parameters:

<CircuitModel> FIMPort | SL | SC | SG

Possible circuit models (character data); see Table 7-8.

Example: See `CALCulate<Ch>:TRANsform:VNETworks:GLOop:DEEMbedding<group>[:STATe]` on page 1139

Manual operation: See "Network" on page 755

CALCulate<Ch>:TRANsform:VNETworks:GLOop:EMBedding<group>:PARameters:C <CircuitModel>, <Capacitance>
CALCulate<Ch>:TRANsform:VNETworks:GLOop:EMBedding<group>:PARameters:C? <CircuitModel>

Specifies the capacitance value C in the different circuit models for ground loop embedding.

Suffix:

<Ch> Channel number.

<group> Port group (DUT) number.
If multiple port groups are configured (see `SOURce<Ch>:GROup<Grp>:PPORts`) and `CALCulate<Ch>:TRANsform:VNETworks:GLOop:GROup` is `ON`, then each port group can have its own de-/embedding models.

Parameters:

<Capacitance> Capacitance C for ground loop embedding.

Range: -1mF to 1 mF.
Increment: 1 fF (1E-15 F)
*RST: 1 pF (1E-12 F)
Default unit: F

Parameters for setting and query:

<CircuitModel> SC | SG

Possible circuit models (character data); see Table 7-8.

Example: `*RST; :CALC:TRAN:VNET:GLO:EMB:PAR:C? SC`
Query the default capacitance for ground loop embedding. The response is 1E-012 (1 pF).
`CALC:TRAN:VNET:GLO:EMB:PAR:C SC, 2.2E-12`
Increase the capacitance to 2.2 pF.

Manual operation: See "Network" on page 755

CALCulate<Ch>:TRANsform:VNETworks:GLOop:EMBedding<group>:PARameters:G <CircuitModel>, <Conductance>
CALCulate<Ch>:TRANsform:VNETworks:GLOop:EMBedding<group>:PARameters:G? <CircuitModel>

Specifies the conductance value G in the different circuit models for ground loop embedding.

Suffix:
<Ch> Channel number.

<group> Port group (DUT) number.
If multiple port groups are configured (see `SOURce<Ch>:GROup<Grp>:PPORts`) and `CALCulate<Ch>:TRANsform:VNETworks:GLOop:GROup` is `ON`, then each port group can have its own de-/embedding models.

Parameters:
<Conductance> Conductance G for the specified circuit model.

Range: -1kS to 1 kS.
Increment: 1 pS (1E-12 S)
*RST: 0 S
Default unit: Siemens (SI unit symbol: S)

Parameters for setting and query:
<CircuitModel> SG

Possible circuit models (character data); see Table 7-8.

Manual operation: See "Network" on page 755

CALCulate<Ch>:TRANsform:VNETworks:GLOop:EMBedding<group>:PARameters:L <CircuitModel>, <Inductance>
CALCulate<Ch>:TRANsform:VNETworks:GLOop:EMBedding<group>:PARameters:L? <CircuitModel>

Specifies the inductance value in the different circuit models for ground loop embedding.

Suffix:
<Ch> Channel number.

<group> Port group (DUT) number.
If multiple port groups are configured (see SOURce<Ch>:GROup<Grp>:PPORts) and CALCulate<Ch>:TRANsform:VNETworks:GLOop:GROup is ON, then each port group can have its own de-/embedding models.

Parameters:

<Inductance> Inductance L for ground loop embedding.

Range: -1H to 1 H.
Increment: 1 pH (1E-12 H)
*RST: 1 nH (1E-9 H)
Default unit: H

Parameters for setting and query:

<CircuitModel> SL

Possible circuit models (character data); see Table 7-8.

Example: `*RST; :CALC:TRAN:VNET:GLO:EMB:PAR:L? SL`
Query the default inductance for ground loop embedding. The response is 1E-009 (1 nH).
`CALC:TRAN:VNET:GLO:EMB:PAR:L SL, 2.2E-9`
Increase the inductance to 2.2 nH.

Manual operation: See "Network" on page 755

CALCulate<Ch>:TRANsform:VNETworks:GLOop:EMBedding<group>:PARameters:R <CircuitModel>, <Resistance>
CALCulate<Ch>:TRANsform:VNETworks:GLOop:EMBedding<group>:PARameters:R? <CircuitModel>

Specifies the resistance value R in the different circuit models for ground loop embedding.

Suffix:

<Ch> Channel number.

<group> Port group (DUT) number.
If multiple port groups are configured (see SOURce<Ch>:GROup<Grp>:PPORts) and CALCulate<Ch>:TRANsform:VNETworks:GLOop:GROup is ON, then each port group can have its own de-/embedding models.

Parameters:

<Resistance> Resistance R for the specified circuit model.

Range: -10 MΩ to 10 MΩ
Increment: 1 mΩ
*RST: 0 Ω if the resistance is connected in series with an inductance. 10 MΩ if the resistance is connected in parallel with a capacitance)
Default unit: Ω

Parameters for setting and query:

<CircuitModel> SL | SC

Possible circuit models (character data); see Table 7-8.

Example: `*RST; :CALC:TRAN:VNET:GLO:EMB:PAR:R? SC; R? SL`
Query the default resistances for ground loop embedding. The response is 10000000; 0.
`CALC:TRAN:VNET:GLO:EMB:PAR:R SC, 2.2E+3`
Increase the resistance for the Shunt C model to 2.2 kΩ.

Manual operation: See "Network" on page 755

CALCulate<Ch>:TRANsform:VNETworks:GLOop:EMBedding<group>[:STATe] <Boolean>

Enables or disables the embedding function for ground loops. It is allowed to change the circuit model and its parameters while embedding is enabled.

Suffix:

<Ch> Channel number.

<group> Port group (DUT) number.
If multiple port groups are configured (see `SOURce<Ch>:GROup<Grp>:PPORts`) and `CALCulate<Ch>:TRANsform:VNETworks:GLOop:GROup` is `ON`, then each port group can have its own de-/embedding models.

Parameters:

<Boolean> ON - Embedding active
OFF - Embedding inactive

*RST: OFF

Example: `CALC:TRAN:VNET:GLO:EMB:TND SL`
Select the Shunt L circuit model for embedding.
`CALC:TRAN:VNET:GLO:EMB:PAR:R SL, 2.2E+3; :CALC:TRAN:VNET:GLO:EMB ON`
Increase the resistance for the Shunt L circuit model to 2.2 kΩ and enable embedding.

Manual operation: See "Active" on page 712

CALCulate<Ch>:TRANsform:VNETworks:GLOop:EMBedding<group>:TNDefinition <CircuitModel>

Selects the circuit model for ground loop embedding.

Suffix:

<Ch> Channel number.

<group> Port group (DUT) number.
If multiple port groups are configured (see SOURce<Ch>:GROup<Grp>:PPORts) and CALCulate<Ch>:TRANsform:VNETworks:GLOop:GROup is ON, then each port group can have its own de-/embedding models.

Parameters:
<CircuitModel> FIMPort | SL | SC | SG
Possible circuit models (character data); see Table 7-8.
*RST: FIMPort

Example: See CALCulate<Ch>:TRANsform:VNETworks:GLOop:EMBedding<group>[:STATe] on page 1143

Manual operation: See "Network" on page 755

CALCulate<Ch>:TRANsform:VNETworks:GLOop:GROup <Boolean>

If set to ON (1), each port group (defined using SOURce<Ch>:GROup<Grp>:PPORts) has its own ground loop embedding and deembedding network. Otherwise the same embedding network and deembedding network is used for all active ports.

See the CALCulate<Ch>:TRANsform:VNETworks:GLOop:DEEMbedding<group>:... and CALCulate<Ch>:TRANsform:VNETworks:GLOop:EMBedding<group>:... commands.

Suffix:
<Ch> Channel number

Parameters:
<Boolean> Range: ON (1) | OFF (0)
*RST: OFF (0)

Manual operation: See "Ground Loop per Port Group" on page 756

CALCulate<Ch>:TRANsform:VNETworks:PPAir:DEEMbedding<ListId>:DEFine <PP_First>,<PP_Second>,<PP_First>,...

Creates one or more port pairs for port set deembedding. The command can be used repeatedly to extend or (partially) overwrite the list of port sets for deembedding.

See CALCulate<Ch>:TRANsform:VNETworks:PSET:DEEMbedding<ListId>:DEFine on page 1154 for general port set definition.

Suffix:
<Ch> Channel number

<ListId> Index of the defined port set within the channel's overall list of port sets for deembedding.
Port sets for deembedding must be numbered consecutively, i.e. port set <ListId> can only be created if port set <ListId>-1 already exists. If several port pairs are specified, <ListId> is the number of the first port pair to be created.

Parameters:

<PP_First>, <PP_Second>, <PP_First>, ... Sequence of port pairs, each one consisting of two different ports.
The port pairs don't have to be disjoint.

Example: See `CALCulate<Ch>:TRANsform:VNETworks:PPAir:DEEMbedding<ListId>[:STATe]`

Usage: Setting only

Manual operation: See "Add / Delete" on page 709

CALCulate<Ch>:TRANsform:VNETworks:PPAir:DEEMbedding:DELete

Deletes all port sets (including port pairs) previously defined for deembedding.

Suffix:

<Ch> Channel number

Example: See `CALCulate<Ch>:TRANsform:VNETworks:PPAir:DEEMbedding<ListId>[:STATe]`

Usage: Event

Manual operation: See "Add / Delete" on page 709

CALCulate<Ch>:TRANsform:VNETworks:PPAir:DEEMbedding<ListId>:PARameters:C<1|2|3> <CircuitModel>, <Capacitance>
CALCulate<Ch>:TRANsform:VNETworks:PPAir:DEEMbedding<ListId>:PARameters:C<1|2|3>? <CircuitModel>

Specifies the capacitance value C<i> in the different lumped circuit models for port pair deembedding.

In the query form, the <Capacitance> parameter must be omitted. The command returns the capacitance value for the specified circuit model.

Suffix:

<Ch> Channel number

<ListId> Index of the affected port pair (see `CALCulate<Ch>:TRANsform:VNETworks:PPAir:DEEMbedding<ListId>:DEFine`)

<1|2|3> Index i of the capacitance C<i> in the related lumped circuit model. If unspecified the numeric suffix is set to 1.

Parameters:

<Capacitance> Capacitance Ci

Range: -1 mF to 1 mF
Increment: 1 fF (1E-15 F)
*RST: 1 pF (1E-12 F)
Default unit: F

Parameters for setting and query:

<CircuitModel> STSC | SCST | CSSL | LSSC | CSSC | SLCS | SCLS | SCCS | STSG | SGST | GSSL | LSSG | GSSG | SLGS | SGLS | SGGS

Circuit model whose capacitance C<i> shall be set, see Table 7-6

Example: See `CALCulate<Ch>:TRANsform:VNETworks:PPAir:DEEMbedding<ListId>[:STATe]`

Manual operation: See "Network" on page 750

CALCulate<Ch>:TRANsform:VNETworks:PPAir:DEEMbedding<ListId>:PARameters:G<1|2|3> <CircuitModel>, <Conductance>
CALCulate<Ch>:TRANsform:VNETworks:PPAir:DEEMbedding<ListId>:PARameters:G<1|2|3>? <CircuitModel>

Specifies the conductance value G<i> in the different lumped circuit models for port pair deembedding.

In the query form, the <Conductance> parameter must be omitted. The command returns the conductance value for the specified circuit model.

Suffix:

<Ch> Channel number

<ListId> Index of the affected port pair (see `CALCulate<Ch>:TRANsform:VNETworks:PPAir:DEEMbedding<ListId>:DEFine`)

<1|2|3> Index i of the conductance G<i> in the related lumped circuit model. If unspecified the numeric suffix is set to 1.

Parameters:

<Conductance> Conductance G<i> for the specified circuit model.

Range: -1kS to 1 kS.
Increment: 1 nS (1E-9 S)
*RST: 0 S
Default unit: Siemens (SI unit symbol: S)

Parameters for setting and query:

<CircuitModel> STSG | SGST | GSSL | LSSG | GSSG | SLGS | SGLS | SGGS

Circuit model whose conductance G<i> shall be set, see Table 7-6

Manual operation: See "Network" on page 750

CALCulate<Ch>:TRANsform:VNETworks:PPAir:DEEMbedding<ListId>:
PARameters:L<1|2|3> <CircuitModel>, <Inductance>
CALCulate<Ch>:TRANsform:VNETworks:PPAir:DEEMbedding<ListId>:
PARameters:L<1|2|3>? <CircuitModel>

Specifies the inductance value L<i> in the different lumped circuit models for port pair deembedding.

In the query form, the <Inductance> parameter must be omitted. The command returns the inductance value for the specified circuit model.

Suffix:

<Ch> Channel number

<ListId> Index of the affected port pair (see `CALCulate<Ch>:TRANsform:VNETworks:PPAir:DEEMbedding<ListId>:DEFine`)

<1|2|3> Index i of the inductance L<i> in the related lumped circuit model. If unspecified the numeric suffix is set to 1.

Parameters:

<Inductance> Inductance L<i>

Range: -1 H to 1 H.
Increment: 1 pH (1E-12 H)
*RST: 1 nH (1E-9 H)
Default unit: H

Parameters for setting and query:

<CircuitModel> STSL | SLST | CSSL | LSSC | LSSL | SLCS | SCLS | SLLS | GSSL | LSSG | SLGS | SGLS

Circuit model whose inductance L<i> shall be set, see Table 7-6

Example: See `CALCulate<Ch>:TRANsform:VNETworks:PPAir:DEEMbedding<ListId>[:STATe]`

Manual operation: See "Network" on page 750

CALCulate<Ch>:TRANsform:VNETworks:PPAir:DEEMbedding<ListId>:
PARameters:R<1|2|3> <CircuitModel>[, <Resistance>]

Specifies the resistance value R<i> in the different lumped circuit models for port pair deembedding.

In the query form, the <Resistance> parameter must be omitted. The command returns the resistance value for the specified circuit model.

Suffix:

<Ch> Channel number

<ListId> Index of the affected port pair (see `CALCulate<Ch>:TRANsform:VNETworks:PPAir:DEEMbedding<ListId>:DEFine`)

<1|2|3> Index i of the resistance R<i> in the related lumped circuit model. If unspecified the numeric suffix is set to 1.

Parameters:

<CircuitModel> STSL | STSC | SLST | SCST | CSSL | LSSC | CSSC | LSSL | SLCS | SCLS | SCCS | SLLS | GSSL | LSSG | SLGS | SGLS

Circuit model whose resistance R<i> shall be set, see Table 7-6

<Resistance> Resistance R<i> for the specified circuit model.

Range: -10 MΩ to 10 MΩ
Increment: 1 mΩ
*RST: 0 Ω if the resistance is connected in series with an inductance. 10 MΩ if the resistance is connected in parallel with a capacitance)
Default unit: Ohm

Example: See `CALCulate<Ch>:TRANsform:VNETworks:PPAir:DEEMbedding<ListId>[:STATe]`

Manual operation: See "Network" on page 750

CALCulate<Ch>:TRANsform:VNETworks:PPAir:DEEMbedding<ListId>[:STATe]
<arg0>

Enables or disables the deembedding function for port set (or port pair) <ListId>. It is allowed to change the deembedding network while embedding is enabled.

Suffix:

<Ch> Channel number

<ListId> Index of the affected port set

Parameters:

<arg0> **OFF** (0): Deembedding inactive
ON (1): Deembedding active

*RST: OFF (0)

Example:

```
*RST; CALC:TRAN:VNET:PPA:DEEM:DEF 1,2,3,4
```
Define a port pair configuration with port pairs (1,2) and (3,4).
```
CALC:TRAN:VNET:PPA:DEEM1:TND CSSL
```
Select the Serial Cs, shunt L circuit model for the first port pair.
```
CALC:TRAN:VNET:PPA:DEEM1:PAR:
R3 CSSL, 2.2E+3; CALC:TRAN:VNET:PPA:DEEM1 ON
```
Increase the resistance R3 for the Serial Cs, shunt L circuit model to 2.2 kΩ and enable deembedding.
```
CALC:TRAN:VNET:PPA:DEEM:DEL
```
Delete the port pair configuration.

Manual operation: See "Active" on page 710

CALCulate<Ch>:TRANsform:VNETworks:PPAir:DEEMbedding<ListId>:TNDefinition <arg0>

Selects the circuit model for port pair deembedding.

Suffix:

<Ch> Channel number

<ListId> Index of the affected port pair (see `CALCulate<Ch>:TRANsform:VNETworks:PPAir:DEEMbedding<ListId>:DEFine`)

Parameters:

<arg0> FIMPort | STSL | STSC | SLST | SCST | CSSL | LSSC | CSSC | LSSL | SLCS | SCLS | SCCS | SLLS | STSG | SGST | GSSL | LSSG | GSSG | SLGS | SGLS | SGGS

Circuit model to be used for the addressed port pair, see Table 7-6

Example: See `CALCulate<Ch>:TRANsform:VNETworks:PPAir:DEEMbedding<ListId>[:STATe]`

Manual operation: See "Network" on page 750

CALCulate<Ch>:TRANsform:VNETworks:PPAir:EMBedding<ListId>:DEFine <PP_First>,<PP_Second>,<PP_First>,...

Creates one or more port pairs for port set embedding. The command can be used repeatedly to extend or (partially) overwrite the list of port sets for embedding.

See `CALCulate<Ch>:TRANsform:VNETworks:PSET:EMBedding<ListId>:DEFine` for general port set definition.

Suffix:

<Ch> Channel number

<ListId> Index of the defined port set within the channel's overall list of port sets for embedding.
Port sets for embedding must be numbered consecutively, i.e. port set <ListId> can only be created if port set <ListId>-1 already exists. If several port pairs are specified, <ListId> is the number of the first port pair to be created.

Parameters:

<PP_First>, <PP_Second>, <PP_First>, ... Sequence of port pairs, each one consisting of two different ports.
The port pairs don't have to be disjoint.

Example: See `CALCulate<Ch>:TRANsform:VNETworks:PPAir:EMBedding<ListId>[:STATe]`

Usage: Setting only

Manual operation: See "Add / Delete" on page 709

CALCulate<Ch>:TRANsform:VNETworks:PPAir:EMBedding<ListId>:DELete

Deletes all port sets (including port pairs) previously defined for embedding.

Suffix:
<Ch> Channel number

<ListId> This suffix is ignored

Example: See `CALCulate<Ch>:TRANsform:VNETworks:PPAir:EMBedding<ListId>[:STATe]`

Usage: Event

Manual operation: See "Add / Delete" on page 709

CALCulate<Ch>:TRANsform:VNETworks:PPAir:EMBedding<ListId>:PARameters:C<1|2|3> <CircuitModel>, <Capacitance>
CALCulate<Ch>:TRANsform:VNETworks:PPAir:EMBedding<ListId>:PARameters:C<1|2|3>? <CircuitModel>

Specifies the capacitance value C<i> in the different lumped circuit models for port pair embedding.

In the query form, the <Capacitance> parameter must be omitted. The command returns the capacitance value for the specified circuit model.

Suffix:
<Ch> Channel number

<ListId> Index of the affected port pair (see `CALCulate<Ch>:TRANsform:VNETworks:PPAir:DEEMbedding<ListId>:DEFine`)

<1|2|3> Index i of the capacitance C<i> in the related lumped circuit model. If unspecified the numeric suffix is set to 1.

Parameters:
<Capacitance> Capacitance Ci

Range: -1 mF to 1 mF
Increment: 1 fF (1E-15 F)
*RST: 1 pF (1E-12 F)
Default unit: F

Parameters for setting and query:
<CircuitModel> STSC | SCST | CSSL | LSSC | CSSC | SLCS | SCLS | SCCS | STSG | SGST | GSSL | LSSG | GSSG | SLGS | SGLS | SGGS

Circuit model whose capacitance C<i> shall be set, see Table 7-6

Example: See `CALCulate<Ch>:TRANsform:VNETworks:PPAir:EMBedding<ListId>[:STATe]`

Manual operation: See "Network" on page 750

CALCulate<Ch>:TRANsform:VNETworks:PPAir:EMBedding<ListId>:PARameters:L<1|2|3> <arg0>, <Inductance>
CALCulate<Ch>:TRANsform:VNETworks:PPAir:EMBedding<ListId>:PARameters:L<1|2|3>? <arg0>

Specifies the inductance value L<i> in the different lumped circuit models for port pair embedding.

In the query form, the <Inductance> parameter must be omitted. The command returns the inductance value for the specified circuit model.

Suffix:
<Ch> Channel number

<ListId> Index of the affected port pair (see `CALCulate<Ch>:TRANsform:VNETworks:PPAir:DEEMbedding<ListId>:DEFine`)

<1|2|3> Index i of the inductance L<i> in the related lumped circuit model. If unspecified the numeric suffix is set to 1.

Parameters:
<Inductance> Inductance L<i>

Range: -1 H to 1 H.
Increment: 1 pH (1E-12 H)
*RST: 1 nH (1E-9 H)
Default unit: H

Parameters for setting and query:
<arg0> STSL | SLST | CSSL | LSSC | LSSL | SLCS | SCLS | SLLS | GSSL | LSSG | SLGS | SGLS

Circuit model whose inductance L<i> shall be set, see Table 7-6

Example: See `CALCulate<Ch>:TRANsform:VNETworks:PPAir:EMBedding<ListId>[:STATe]`

Manual operation: See "Network" on page 750

CALCulate<Ch>:TRANsform:VNETworks:PPAir:EMBedding<ListId>:PARameters:R<1|2|3> <arg0>, <Resistance>
CALCulate<Ch>:TRANsform:VNETworks:PPAir:EMBedding<ListId>:PARameters:R<1|2|3>? <arg0>

Specifies the resistance value R<i> in the different lumped circuit models for port pair embedding.

In the query form, the <Resistance> parameter must be omitted. The command returns the resistance value for the specified circuit model.

Suffix:
<Ch> Channel number

<ListId> Index of the affected port pair (see CALCulate<Ch>:TRANsform:VNETworks:PPAir:DEEMbedding<ListId>:DEFine)

<1|2|3> Index i of the resistance R<i> in the related lumped circuit model. If unspecified the numeric suffix is set to 1.

Parameters:

<Resistance> Resistance R<i> for the specified circuit model.

Range: -10 MΩ to 10 MΩ
Increment: 1 mΩ
*RST: 0 Ω if the resistance is connected in series with an inductance. 10 MΩ if the resistance is connected in parallel with a capacitance)
Default unit: Ohm

Parameters for setting and query:

<arg0> STSL | STSC | SLST | SCST | CSSL | LSSC | CSSC | LSSL | SLCS | SCLS | SCCS | SLLS | GSSL | LSSG | SLGS | SGLS

Circuit model whose resistance R<i> shall be set, see Table 7-6

Example: See CALCulate<Ch>:TRANsform:VNETworks:PPAir:EMBedding<ListId>[:STATe]

Manual operation: See "Network" on page 750

CALCulate<Ch>:TRANsform:VNETworks:PPAir:EMBedding<ListId>:PARameters:G<1|2|3> <arg0>, <Conductance>
CALCulate<Ch>:TRANsform:VNETworks:PPAir:EMBedding<ListId>:PARameters:G<1|2|3>? <arg0>

Specifies the conductance value G<i> in the different lumped circuit models for port pair embedding.

In the query form, the <Conductance> parameter must be omitted. The command returns the conductance value for the specified circuit model.

Suffix:

<Ch> Channel number

<ListId> Index of the affected port pair (see CALCulate<Ch>:TRANsform:VNETworks:PPAir:DEEMbedding<ListId>:DEFine)

<1|2|3> Index i of the conductance G<i> in the related lumped circuit model. If unspecified the numeric suffix is set to 1.

Parameters:

<Conductance> Conductance G<i> for the specified circuit model.

Range: -1kS to 1 kS.
Increment: 1 pS (1E-12 S)
*RST: 0 S
Default unit: Siemens (SI unit symbol: S)

Parameters for setting and query:

<arg0> STSG | SGST | GSSL | LSSG | GSSG | SLGS | SGLS | SGGS

Circuit model whose conductance G<i> shall be set, see Table 7-6

Manual operation: See "Network" on page 750

CALCulate<Ch>:TRANsform:VNETworks:PPAir:EMBedding<ListId>[:STATe]
<Boolean>

Enables or disables the embedding function for **port set** <ListId>. It is allowed to change the embedding network while embedding is enabled.

Suffix:

<Ch> Channel number

<ListId> Index of the affected **port set** (see `CALCulate<Ch>:TRANsform:VNETworks:PPAir:EMBedding<ListId>:DEFine`)

Parameters:

<Boolean> **OFF** (0): Embedding inactive
ON (1): Embedding active

*RST: OFF (0)

Example:

```
*RST; CALC:TRAN:VNET:PPA:EMB:DEF 1,2,3,4
```
Define a port pair configuration with port pairs (1,2) and (3,4).
```
CALC:TRAN:VNET:PPA:EMB1:TND CSSL
```
Select the Serial Cs, shunt L circuit model for the first port pair.
```
CALC:TRAN:VNET:PPA:EMB1:PAR:
R3 CSSL, 2.2E+3; CALC:TRAN:VNET:PPA:EMB1 ON
```
Increase the resistance R3 for the Serial Cs, shunt L circuit model to 2.2 kΩ and enable deembedding.
```
CALC:TRAN:VNET:PPA:EMB:DEL
```
Delete the port pair configuration.

Manual operation: See "Active" on page 710

CALCulate<Ch>:TRANsform:VNETworks:PPAir:EMBedding<ListId>:TNDefinition
<CircuitModel>

Selects the circuit model for port pair embedding.

Suffix:

<Ch> Channel number

<ListId> Index of the affected port pair (see `CALCulate<Ch>:TRANsform:VNETworks:PPAir:EMBedding<ListId>:DEFine`)

Parameters:

<CircuitModel> FIMPort | STSL | STSC | SLST | SCST | CSSL | LSSC | CSSC | LSSL | SLCS | SCLS | SCCS | SLLS | STSG | SGST | GSSL | LSSG | GSSG | SLGS | SGLS | SGGS

Circuit model to be used for the addressed port pair, see Table 7-6

Example: See `CALCulate<Ch>:TRANsform:VNETworks:PPAir:EMBedding<ListId>[:STATe]`

Manual operation: See "Network" on page 750

CALCulate<Ch>:TRANsform:VNETworks:PSET:DEEMbedding<ListId>:DEFine
<Port1>, <Port2>[, <Port3>[, <Port4>]]
CALCulate<Ch>:TRANsform:VNETworks:PSET:EMBedding<ListId>:DEFine
<Port1>, <Port2>, ...

Defines port set <ListId> for port set deembedding|embedding.

Note that **port pairs** (i.e. 2-element port sets) can also be created using `CALCulate<Ch>:TRANsform:VNETworks:PPAir:DEEMbedding<ListId>:DEFine` / `CALCulate<Ch>:TRANsform:VNETworks:PPAir:EMBedding<ListId>:DEFine`.

Use `CALCulate<Ch>:TRANsform:VNETworks:PPAir:DEEMbedding:DELete` / `CALCulate<Ch>:TRANsform:VNETworks:PPAir:EMBedding<ListId>:DELete` on page 1150 to delete all port sets (including port pairs).

Suffix:

<Ch> Channel number

<ListId> Index of the defined port set within the channel's overall list of port sets for deembedding/embedding.

Parameters:

<Port1>, <Port2>, ... A port set consist of two or more (different) ports.
A port can be an element of multiple port sets.

Manual operation: See "Add / Delete" on page 709

CALCulate<Ch>:TRANsform:VNETworks:SENDed:DEEMbedding<PhyPt>:IGNore
<IgnoreReflectionType>

For single-ended deembedding using Touchstone files (`MMEMory:LOAD:VNETworks<Ch>:SENDed:DEEMbedding<PhyPt>`), you can either use the reflection S-parameters of the `s2p` file or, or ignore them and assume zero reflection.

Suffix:

<Ch> Channel number

<PhyPt> Physical port number

Parameters:

<IgnoreReflectionType> NONE | ZERo_snn

NONE
S_{nn} from `s2p` file.

ZERo_snn
Assume $S_{11} = S_{22} = 0$

Manual operation: See "Modify" on page 708

CALCulate<Ch>:TRANsform:VNETworks:SENDed:DEEMbedding<PhyPt>:PARameters:C<Cmp> <CircuitModel>, <Capacitance>
CALCulate<Ch>:TRANsform:VNETworks:SENDed:DEEMbedding<PhyPt>:PARameters:C<Cmp>? <CircuitModel>

Specifies the capacitance value C<Cmp> in the different circuit models for single ended port embedding.

In the query form, the <Capacitance> parameter must be omitted. The command returns the capacitance value for the specified circuit model.

Suffix:
<Ch> Channel number

<PhyPt> Physical port number

<Cmp> Number of capacitance in circuit model. The total number of capacitances depends on the selected circuit model.

Parameters:
<Capacitance> Capacitance C<Cmp> for the specified circuit model.

Range: -1mF to 1 mF.
Increment: 1 fF (1E-15 F)
*RST: 1 pF (1E-12 F)
Default unit: F

Parameters for setting and query:
<CircuitModel> CSL | LSC | CSC | SLC | SCL | SCC | SHLC | GSL | LSG | GSG | SLG | SGL | SGG

Possible circuit models (character data); see Table 7-5.

Example: `*RST; :CALC:TRAN:VNET:SEND:DEEM:PAR:C2? CSC`
Query the default capacitance C2 for the Serial C, shunt C circuit model. The response is 1E-012 (1 pF).
`CALC:TRAN:VNET:SEND:DEEM:PAR:C2 CSC, 2.2E-12`
Increase the capacitance to 2.2 pF.

Manual operation: See "Network" on page 747

CALCulate<Ch>:TRANsform:VNETworks:SENDed:DEEMbedding<PhyPt>:PARameters:DATA <Interchange>, <arg1>

Defines an embedding network for a single-ended port based on the given S-Parameter traces.

Use

- `CALCulate<Ch>:TRANsform:VNETworks:SENDed:DEEMbedding<PhyPt>:TNDefinition` to select the adequate circuit model **before** executing this command.
- `MMEMory:LOAD:VNETworks<Ch>:SENDed:DEEMbedding<PhyPt>` to load circuit data from a Touchstone file located at the R&S ZNB's file system.

Suffix:

<Ch> Channel number

<PhyPt> Physical port number

Setting parameters:

<Interchange> FPORts | IPORts | SGATes

FPORts (or omitted)
Standard port sequence (network port 1 towards VNA, network port 2 towards DUT)

IPORts | SGATes
Inverted port sequence (network port 2 towards VNA, network port 1 towards DUT)

<arg1> <block_data>

Content of a Touchstone file (`*.s2p`) in IEEE488.2 Block data format.

Usage: Setting only

CALCulate<Ch>:TRANsform:VNETworks:SENDed:DEEMbedding<PhyPt>:PARameters:G<Cmp> <CircuitModel>, <Conductance>
CALCulate<Ch>:TRANsform:VNETworks:SENDed:DEEMbedding<PhyPt>:PARameters:G<Cmp>? <CircuitModel>

Specifies the conductance value G in the "shunt L, shunt C" circuit model for single ended port deembedding.

In the query form, the <Conductance> parameter must be omitted.

Suffix:

<Ch> Channel number

<PhyPt> Physical port number

<Cmp> Number of the conductance component in the circuit model. The total number of conductances depends on the selected circuit model.

Parameters:

<Conductance> Conductance G<Cmp> for the specified circuit model.

Range: -1kS to 1 kS.
Increment: 1 nS (1E-9 S)
*RST: 0 S
Default unit: Siemens (SI unit symbol: S)

Parameters for setting and query:

<CircuitModel> SHLC

Circuit model whose conductance G<Cmp> shall be set, see Table 7-5.

Example: `*RST; :CALC:TRAN:VNET:SEND:DEEM:PAR:L? SHLC; R? SHLC; C? SHLC; G? SHLC`
Query the default component values for the "shunt L, shunt C" circuit model.
`CALC:TRAN:VNET:SEND:DEEM:PAR:G SHLC, 1`
Increase the conductance G to 1 Siemens.

Manual operation: See "Network" on page 747

CALCulate<Ch>:TRANsform:VNETworks:SENDed:DEEMbedding<PhyPt>:PARameters:L<Cmp> <CircuitModel>, <Inductance>
CALCulate<Ch>:TRANsform:VNETworks:SENDed:DEEMbedding<PhyPt>:PARameters:L<Cmp>? <CircuitModel>

Specifies the inductance value L<Cmp> in the different circuit models for single ended port deembedding.

In the query form, the <Inductance> parameter must be omitted. The command returns the inductance value for the specified circuit model.

Suffix:

<Ch> Channel number

<PhyPt> Physical port number

<Cmp> Number of inductance in circuit model. The total number of inductances depends on the selected circuit model.

Parameters:

<Inductance> Inductance L<Cmp> for the specified circuit model.

Range: -1H to 1 H.
Increment: 1 pH (1E-12 H)
*RST: 1 nH (1E-9 H)
Default unit: H

Parameters for setting and query:

<CircuitModel> CSL | LSC | LSL | SLC | SCL | SLL | SHLC | GSL | LSG | SLG | SGL

Possible circuit models (character data); see Table 7-5.

Example: `*RST; :CALC:TRAN:VNET:SEND:DEEM:PAR:L1? SLL`
Query the default inductance L1 for the Shunt L, serial L circuit model. The response is 1E-009 (1 nH).
`CALC:TRAN:VNET:SEND:DEEM:PAR:L1 SLL, 2.2E-9`
Increase the inductance to 2.2 nH.

Manual operation: See "Network" on page 747

CALCulate<Ch>:TRANsform:VNETworks:SENDed:DEEMbedding<PhyPt>:PARameters:R<Cmp> <CircuitModel>, <Resistance>
CALCulate<Ch>:TRANsform:VNETworks:SENDed:DEEMbedding<PhyPt>:PARameters:R<Cmp>? <CircuitModel>

Specifies the resistance value R<Cmp> in the different circuit models for single ended port deembedding.

In the query form, the <Resistance> parameter must be omitted. The command returns the resistance value for the specified circuit model.

Suffix:
<Ch> Channel number

<PhyPt> Physical port number

<Cmp> Number of resistance in circuit model. The total number of resistances depends on the selected circuit model.

Parameters:
<Resistance> Resistance R<Cmp> for the specified circuit model.

Range: -10 MΩ to 10 MΩ.
Increment: 1 mΩ (1E-3 Ω)
*RST: 0 Ω for all resistances connected in series with an inductance. 10 MΩ for all resistances connected in parallel with a capacitance
Default unit: Ohm

Parameters for setting and query:
<CircuitModel> CSL | LSC | CSC | LSL | SLC | SCL | SCC | SLL | SHLC | GSL | LSG | SLG | SGL

Possible circuit models (character data); see Table 7-5.

Example: `*RST; :CALC:TRAN:VNET:SEND:DEEM:PAR:R1? CSL; R2? CSL`
Query the default resistances for the Serial C, shunt L circuit model. The response is 10000000; 0.
`CALC:TRAN:VNET:SEND:DEEM:PAR:R2 CSL, 2.2E+3`
Increase the resistance R2 to 2.2 kΩ.

Manual operation: See "Network" on page 747

CALCulate<Ch>:TRANsform:VNETworks:SENDed:DEEMbedding<PhyPt>[:STATe] <Boolean>

Enables or disables the deembedding function for single ended ports. It is allowed to change the circuit model and its parameters while deembedding is enabled.

Suffix:
<Ch> Channel number

<PhyPt> Physical port number

Parameters:

<Boolean> ON - deembedding active
OFF - deembedding inactive
*RST: OFF

Example: `CALC:TRAN:VNET:SEND:DEEM:TND CSL`
Select the Serial C, shunt L circuit model for deembedding.
`CALC:TRAN:VNET:SEND:DEEM:PAR:R2 CSL, 2.2E+3; :`
`CALC:TRAN:VNET:SEND:DEEM ON`
Increase the resistance R2 for the Serial C, shunt L circuit model to 2.2 kΩ and enable deembedding.

Manual operation: See "Active" on page 708

CALCulate<Ch>:TRANsform:VNETworks:SENDed:DEEMbedding<PhyPt>:TNDefinition <CircuitModel>

Selects the circuit model for single ended port deembedding.

Suffix:

<Ch> Channel number

<PhyPt> Physical port number

Parameters:

<CircuitModel> FIMPort | CSL | LSC | CSC | LSL | SLC | SCL | SCC | SLL | SHLC | GSL | LSG | GSG | SLG | SGL | SGG
Possible circuit models (character data); see Table 7-5
*RST: CSL

Example: See `CALCulate<Ch>:TRANsform:VNETworks:SENDed:DEEMbedding<PhyPt>[:STATe]`

Manual operation: See "Network" on page 747

CALCulate<Ch>:TRANsform:VNETworks:SENDed:EMBedding<PhyPt>:PARameters:C<Cmp> <CircuitModel>, <Capacitance>
CALCulate<Ch>:TRANsform:VNETworks:SENDed:EMBedding<PhyPt>:PARameters:C<Cmp>? <CircuitModel>

Specifies the capacitance value C<Cmp> in the different circuit models for single ended port embedding.

In the query form, the <Capacitance> parameter must be omitted. The command returns the capacitance value for the specified circuit model.

Suffix:

<Ch> Channel number

<PhyPt> Physical port number

<Cmp> Number of capacitance in circuit model. The total number of capacitances depends on the selected circuit model.

Parameters:

<Capacitance> Capacitance C<Cmp> for the specified circuit model.

Range: -1mF to 1 mF.
Increment: 1 fF (1E-15 F)
*RST: 1 pF (1E-12 F)
Default unit: F

Parameters for setting and query:

<CircuitModel> CSL | LSC | CSC | SLC | SCL | SCC | SHLC | GSL | LSG | GSG | SLG | SGL | SGG

Possible circuit models (character data); see Table 7-5.

Example:
```
*RST; :CALC:TRAN:VNET:SEND:EMB:PAR:C2? CSC
```
Query the default capacitance C2 for the Serial C, shunt C circuit model. The response is 1E-012 (1 pF).
```
CALC:TRAN:VNET:SEND:EMB:PAR:C2 CSC, 2.2E-12
```
Increase the capacitance to 2.2 pF.

Manual operation: See "Network" on page 747

CALCulate<Ch>:TRANsform:VNETworks:SENDed:EMBedding<PhyPt>:PARameters:DATA <Interchange>, <arg1>

Defines a dembedding network for a single-ended port based on the given S-Parameter traces.

Use

- `CALCulate<Ch>:TRANsform:VNETworks:SENDed:EMBedding<PhyPt>:TNDefinition` to select the adequate circuit model **before** executing this command.
- `MMEMory:LOAD:VNETworks<Ch>:SENDed:EMBedding<PhyPt>` to load circuit data from a Touchstone file located at the R&S ZNB's file system.

Suffix:

<Ch> Channel number

<PhyPt> Physical port number

Setting parameters:

<Interchange> FPORts | IPORts | SGATes

FPORts (or omitted)
Standard port sequence (network port 1 towards VNA, network port 2 towards DUT)

IPORts | SGATes
Inverted port sequence (network port 2 towards VNA, network port 1 towards DUT)

<arg1> <block_data>

Content of a Touchstone file (`*.s2p` or `*.s4p`) in IEEE488.2 Block data format.

Usage: Setting only

CALCulate<Ch>:TRANsform:VNETworks:SENDed:EMBedding<PhyPt>:PARameters:G<Cmp> <CircuitModel>, <Conductance>
CALCulate<Ch>:TRANsform:VNETworks:SENDed:EMBedding<PhyPt>:PARameters:G<Cmp>? <CircuitModel>

Specifies the conductance value G<Cmp> in the different circuit models for single ended port embedding.

In the query form, the <Conductance> parameter must be omitted. The command returns the conductance value for the specified circuit model.

Suffix:
<Ch> Channel number

<PhyPt> Physical port number

<Cmp> Number of the conductance component in the circuit model. The total number of conductances depends on the selected circuit model.

Parameters:
<Conductance> Conductance G<Cmp> for the specified circuit model.

Range: -1kS to 1 kS.
Increment: 1 pS (1E-12 S)
*RST: 0 S
Default unit: Siemens (SI unit symbol: S)

Parameters for setting and query:
<CircuitModel> GSL | LSG | GSG | SLG | SGL | SGG | SHLC

Circuit model whose conductance G<Cmp> shall be set, see Table 7-5.

Example: `*RST; :CALC:TRAN:VNET:SEND:EMB:PAR:L? SHLC; R? SHLC; C? SHLC; G? SHLC`
Query the default component values for the "shunt L, shunt C" circuit model.
`CALC:TRAN:VNET:SEND:EMB:PAR:G SHLC, 1`
Increase the conductance G to 1 Siemens.

Manual operation: See "Network" on page 747

CALCulate<Ch>:TRANsform:VNETworks:SENDed:EMBedding<PhyPt>:PARameters:L<Cmp> <CircuitModel>, <Inductance>
CALCulate<Ch>:TRANsform:VNETworks:SENDed:EMBedding<PhyPt>:PARameters:L<Cmp>? <CircuitModel>

Specifies the inductance value L<Cmp> in the different circuit models for single ended port embedding.

In the query form, the <Inductance> parameter must be omitted. The command returns the inductance value for the specified circuit model.

Suffix:
<Ch> Channel number

<PhyPt> Physical port number

<Cmp> Number of inductance in circuit model. The total number of inductances depends on the selected circuit model.

Parameters:

<Inductance> Inductance L<Cmp> for the specified circuit model.

Range: -1H to 1 H.
Increment: 1 pH (1E-12 H)
*RST: 1 nH (1E-9 H)
Default unit: H

Parameters for setting and query:

<CircuitModel> CSL | LSC | LSL | SLC | SCL | SLL | SHLC | GSL | LSG | SLG | SGL

Possible circuit models (character data); see Table 7-5.

Example:

```
*RST; :CALC:TRAN:VNET:SEND:EMB:PAR:L1? SLL
```

Query the default inductance L1 for the Shunt L, serial L circuit model. The response is 1E-009 (1 nH).

```
CALC:TRAN:VNET:SEND:EMB:PAR:L1 SLL, 2.2E-9
```

Increase the inductance to 2.2 nH.

Manual operation: See "Network" on page 747

CALCulate<Ch>:TRANsform:VNETworks:SENDed:EMBedding<PhyPt>:PARameters:R<Cmp> <CircuitModel>, <Resistance>
CALCulate<Ch>:TRANsform:VNETworks:SENDed:EMBedding<PhyPt>:PARameters:R<Cmp>? <CircuitModel>

Specifies the resistance value R<Cmp> in the different circuit models for single ended port embedding.

In the query form, the <Resistance> parameter must be omitted. The command returns the resistance value for the specified circuit model.

Suffix:

<Ch> Channel number

<PhyPt> Physical port number

<Cmp> Number of resistance in circuit model. The total number of resistances depends on the selected circuit model.

Parameters:

<Resistance> Resistance R<Cmp> for the specified circuit model.

Range: -10 MΩ to 10 MΩ.
Increment: 1 mΩ (1E-3 Ω)
*RST: 0 Ω for all resistances connected in series with an inductance. 10 MΩ for all resistances connected in parallel with a capacitance
Default unit: Ohm

Parameters for setting and query:

<CircuitModel> CSL | LSC | CSC | LSL | SLC | SCL | SCC | SLL | SHLC | GSL | LSG | SLG | SGL

Possible circuit models (character data); see Table 7-5.

Example: `*RST; :CALC:TRAN:VNET:SEND:EMB:PAR:R1? CSL; R2? CSL`

Query the default resistances for the Serial C, shunt L circuit model. The response is 10000000; 0.

`CALC:TRAN:VNET:SEND:EMB:PAR:R2 CSL, 2.2E+3`

Increase the resistance R2 to 2.2 kΩ.

Manual operation: See "Network" on page 747

CALCulate<Ch>:TRANsform:VNETworks:SENDed:EMBedding<PhyPt>[:STATe] <Boolean>

Enables or disables the embedding function for single ended ports. It is allowed to change the circuit model and its parameters while embedding is enabled.

Suffix:

<Ch> Channel number

<PhyPt> Physical port number

Parameters:

<Boolean> ON - embedding active
OFF - embedding inactive

*RST: OFF

Example: `CALC:TRAN:VNET:SEND:EMB:TND CSL`

Select the Serial C, shunt L circuit model for embedding.

`CALC:TRAN:VNET:SEND:EMB:PAR:R2 CSL, 2.2E+3; :CALC:TRAN:VNET:SEND:EMB ON`

Increase the resistance R2 for the Serial C, shunt L circuit model to 2.2 kΩ and enable embedding.

Manual operation: See "Active" on page 708

CALCulate<Ch>:TRANsform:VNETworks:SENDed:EMBedding<PhyPt>:TNDefinition <CircuitModel>

Selects the circuit model for single ended port embedding.

Suffix:

<Ch> Channel number

<PhyPt> Physical port number

Parameters:

<CircuitModel> FIMPort | CSL | LSC | CSC | LSL | SLC | SCL | SCC | SLL | SHLC | GSL | LSG | GSG | SLG | SGL | SGG

Possible circuit models (character data); see Table 7-5

*RST: CSL

Example: See `CALCulate<Ch>:TRANsform:VNETworks:SENDed:EMBedding<PhyPt>[:STATe]`

Manual operation: See "Network" on page 747

CALCulate<Ch>:TRANsform:VNETworks:WAVes <Boolean>

Defines whether the wave-based, or the classical S parameter-based de-/embedding calculation is used in channel <Ch>.

Suffix:

<Ch> Channel number

Parameters:

<Boolean> **ON (1)**
Wave-based calculation (default)

OFF (0)
S parameter-based calculation

Manual operation: See "Offset > Wave De-/Embed." on page 760

CALCulate:TRANsform... (other)

CALCulate<Chn>:TRANsform:COMPlex <Result>

Converts S-parameters into converted (matched-circuit) Y-parameters or Z-parameters and vice versa, assuming that port no. i is terminated with Z_{0i} so that the three parameter sets are equivalent and the following formulas apply:

$$Z_{ii} = Z_{0i}\frac{1+S_{ii}}{1-S_{ii}}$$

$$Z_{ij} = 2\frac{\sqrt{Z_{0i}Z_{0j}}}{S_{ij}} - (Z_{0i}+Z_{0j}) \quad \text{for } i \neq j$$

$$Y_{ii} = 1/Z_{ij} = \frac{1}{Z_{0i}} \cdot \frac{1-S_{ii}}{1+S_{ii}}$$

$$Y_{ij} = 1/Z_{ij} = \frac{S_{ij}}{2\sqrt{Z_{0i}Z_{0j}} - S_{ij}(Z_{0i}+Z_{0j})} \quad \text{for } i \neq j$$

Suffix:

<Chn> Channel number used to identify the active trace

Parameters:

<Result> S | Y | Z

S-parameters, Y-parameters, Z-parameters

Example: `*RST; CALC:PAR:MEAS 'Trc1'", '"Y-S22'`
Select the converted admittance Y <-- S22 as measurement parameter of the default trace.
`CALC:TRAN:COMP S`
Convert the converted Y-parameter into an S-parameter.

CALCulate<Chn>:TRANsform:IMPedance:RNORmal <Model>

Selects the theory for the renormalization of port impedances. The selection has an impact on the conversion formulas for wave quantities and S-parameters.

Suffix:

<Chn> Channel number used to identify the active trace

Parameters:

<Model> TWAVes | PWAVes

TWAVes - travelling waves
PWAVes - power waves

*RST: TWAVes

Example: See `[SENSe<Ch>:]PORT<PhyPt>:ZREFerence`

Manual operation: See "Renormalization According to Theory of" on page 302

CALCulate<Chn>:TRANsform:PEEL:STATe <Boolean>

If enabled, a peeling algorithm is applied on the time domain reflection (TDR) trace. It reduces distortions of the true impedance values in the timeline caused by multiple reflections due to impedance discontinuities in the signal path through the DUT. The algorithm is lossless, that is, no path losses in the DUT are taken into account.

Suffix:

<Chn> Channel number used to identify the active trace.

Parameters:

<Boolean> 0 − Peeling is deactivated.
1 − Peeling is activated.

Manual operation: See "Peel" on page 358

7.3.1.15 CALCulate:TTIMe:...

Defines the properties and retrieves the results of the rise time measurement provided with the Extended time domain analysis option R&S ZNB-K20.

CALCulate<Chn>:TTIMe:DATA....1166
CALCulate<Chn>:TTIMe:STATe....1166
CALCulate<Chn>:TTIMe:THReshold....1167

CALCulate<Chn>:TTIMe:DATA [<Data>]

The setting defines the content of the rise time info field.

The query gets the results of the rise time measurement.

Note: The interpretation of the result values depends on the current trace's stimulus axis (`CALCulate<Chn>:TRANsform:TIME:XAXis`).

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<Data> **ALL**
Setting: The info field displays extended information
Query: The result consists of 8 numeric values (see table below)

(omitted)
Setting: The info field displays basic information
Query: A single numeric value is returned (see table below)

Options: R&S ZNB-K20

Manual operation: See "Extended Info" on page 403

Table 7-10: Return values

Stimulus axis	<Data> omitted	<Data> = ALL
Time	<rise time>	<rise time>, <rise distance>, <start threshold crossing time>, <start threshold crossing voltage>, <stop threshold crossing time>, <stop threshold crossing voltage>, <start percentage>, <stop percentage>
Distance	<rise distance>	<rise distance>, <rise time>, <start threshold crossing distance>, <start threshold crossing voltage>, <stop threshold crossing distance>, <stop threshold crossing voltage>, <start percentage>, <stop percentage>

CALCulate<Chn>:TTIMe:STATe <Boolean>

Enables/disables the Rise time measurement.

Note: The rise time measurement can only be enabled if the active trace is real (`CALCulate<Chn>:FORMat REAL`), Time Domain is enabled (`CALCulate<Chn>:TRANsform:TIME:STATe ON`), and the Low Pass Step time domain transform is used (`CALCulate<Chn>:TRANsform:TIME[:TYPE] LPASs` and `CALCulate<Chn>:TRANsform:TIME:STIMulus STEP`). The latter requires the stimulus grid to be harmonic, which can be achieved, for example, using `[SENSe<Ch>:]HARMonic:AUTO ON`.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<Boolean> ON – Rise time measurement enabled
OFF – Rise time measurement disabled
*RST: OFF

Options: R&S ZNB-K20

Manual operation: See "Rise Time" on page 402

CALCulate<Chn>:TTIMe:THReshold <ThresholdEnum | LowerThreshold>[, <UpperThreshold>]

Defines the lower/upper threshold for the rise time measurement.

The thresholds can either be specified by enum constants for the standard 10–90% or 20–80% rise times, or as integer percentages.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<ThresholdEnum | LowerThreshold> T1_9: 10–90% rise time
T2_8: 20–80% rise time
<integer>: Lower rise time threshold as integer percentage
*RST: T1_9:

<UpperThreshold> Upper rise time threshold as integer percentage

Options: R&S ZNB-K20

Manual operation: See "Start Cursor / Stop Cursor" on page 402

7.3.1.16 CALCulate... (other)

CALCulate:CLIMits:FAIL?....1168
CALCulate:CLIMits[:STATe]....1168
CALCulate<Chn>:DLINe....1168
CALCulate<Chn>:DLINe:STATe....1169
CALCulate<Chn>:FORMat....1169
CALCulate<Chn>:FORMat:WQUType....1171
CALCulate<Chn>:GDAPerture:SCOunt....1171
CALCulate<Chn>:IAVerage:MODE....1171

CALCulate<Chn>:IAVerage[:STATe] 1172
CALCulate<Chn>:PHOLd 1172
CALCulate:PIPelining:DATA? 1172
CALCulate<Chn>:SMOothing:APERture 1173
CALCulate<Chn>:SMOothing[:STATe] 1174
CALCulate<Chn>:TRACking:MODE 1174
CALCulate<Chn>:TRACking:SWEep:POINt 1174
CALCulate:TDVSwr[:STATe] 1174

CALCulate:CLIMits:FAIL?

Returns a 0 or 1 to indicate whether a global, composite limit check on several traces has failed.

The result is automatically recalculated whenever a relevant setting is changed, i.e. a subsequent query returns the updated limit violation state.

Use `CALCulate:CLIMits[:STATe]` to activate or deactivate the global limit check.

Example: `*RST; CALC:LIM:CONT 1 GHZ, 2 GHZ`
Define an upper limit line segment in the stimulus range between 1 GHz and 2 GHz, using default response values.
`CALC:LIM:STAT ON; FAIL?`
Switch the limit check on and query the result.
`CALC:CLIM:STAT ON; FAIL?`
Activate the composite limit check and check its result. As only one trace is tested, the response is the same as the previous one.

Usage: Query only

Manual operation: See "Global Check" on page 486

CALCulate:CLIMits[:STATe] <Boolean>

Activates or deactivates the global limit check including upper/lower limits and ripple limits.

Use `CALCulate:CLIMits:FAIL?` to query its results.

Parameters:
<Boolean> ON (1) | OFF (0)
*RST: OFF (0)

Manual operation: See "Global Check" on page 486

CALCulate<Chn>:DLINe <Position>

Defines the position (response value) of the horizontal line.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:

<Position> See list of parameters below.

Default unit: NN

Example: `*RST; :CALC:DLIN 10`
Define the position of the horizontal line in the default "dB Mag" diagram at +10 dB.
`CALC:DLIN:STAT ON`
Display the defined horizontal line.

Manual operation: See "Response Value" on page 502

CALCulate<Chn>:DLINe:STATe <Boolean>

Switches the horizontal line on or off.

Suffix:

<Chn> Channel number used to identify the active trace

Parameters:

<Boolean> ON | OFF - horizontal line on or off

*RST: OFF

Example: See `CALCulate<Chn>:DLINe`

Manual operation: See "Show Horiz. Line" on page 502

CALCulate<Chn>:FORMat <Type>

Defines how the measured result at any sweep point is post-processed and presented in the graphical display.

Note: The analyzer allows arbitrary combinations of display formats and measured quantities; see Section 5.3, "Format softtool", on page 438 and `CALCulate<Ch>:PARameter...` commands. Nevertheless, it is advisable to check which display formats are appropriate for an analysis of a particular measured quantity; see Section 4.2.3.3, "Measured quantities and trace formats", on page 116.

Suffix:

<Chn> Channel number used to identify the active trace

Parameters:

<Type> MLINear | MLOGarithmic | PHASe | UPHase | POLar | SMITh | ISMith | GDELay | REAL | IMAGinary | SWR | COMPlex | MAGNitude | LOGarithmic | GDDerivation

See list of parameters below.

*RST: MLOGarithmic

Example: `CALC4:PAR:SDEF 'Ch4Tr1', 'S11'`
Create channel 4 and a trace named `Ch4Tr1` to measure the input reflection coefficient S_{11}. The trace becomes the active trace in channel 4.
`CALC4:FORM MLIN; :DISP:WIND:TRAC2:FEED 'CH4TR1'`
Calculate the magnitude of S_{11} and display it in a linearly scaled Cartesian diagram, assigning the trace number 2.

Manual operation: See "dB Mag" on page 438

Assume that the result at a sweep point is given by the complex quantity z = x + jy. Then the magnitude of z is calculated as:

$|z| = \sqrt{x^2 + y^2}$

In phase notation we have:

$z = |z| \, e^{j \, Phase(z)}$, where Phase(z) = arctan(y/x).

The meaning of the parameters is as follows (see also table in `CALCulate<Chn>:MARKer<Mk>:FORMat` description):

MLINear	Displays |z| in a Cartesian diagram
MLOGarithmic MAGNitude (for compatibility with R&S ZVR analyzers)	Calculates |z| in dB (= 20 log|z|) and displays it in a Cartesian diagram
PHASe	Calculates Phase(z) in the range between -180° and +180° and displays it in a Cartesian diagram
UPHase	Calculates Phase(z) (unwrapped) and displays it in a Cartesian diagram
POLar COMPlex (for compatibility with R&S ZVR analyzers)	Displays z in a polar diagram
SMITh	Displays z in a Smith diagram
ISMith	Displays z in an inverted Smith diagram
GDELay	For frequency sweeps only Calculates the group delay at the related sweep point and displays it in a Cartesian diagram
GDDerivation	Group delay derivation
REAL	Calculates Re(z) = x and displays it in a Cartesian diagram
IMAGinary	Calculates Im(z) = y and displays it in a Cartesian diagram
SWR	Calculates the standing wave ratio (1 + |z|) / (1 - |z|) and displays it in a Cartesian diagram
LOGarithmic	Displays z in a Cartesian diagram with logarithmic scale

CALCulate<Chn>:FORMat:WQUType <Unit>

Selects the physical unit of the displayed trace.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<Unit> POWer | VOLTage
Power or voltage units
*RST: POWer

Example: `CALC4:PAR:SDEF 'Ch4Tr1', 'b1'`
Create channel 4 and a trace named `Ch4Tr1` to measure the wave quantity b_1. The trace becomes the active trace in channel 4.
`CALC4:FORM:WQUT VOLT`
Select voltage units for the created trace (identified by the suffix 4).

Manual operation: See "Show as" on page 308

CALCulate<Chn>:GDAPerture:SCOunt <Steps>

Defines an aperture for the calculation of the group delay as an integer number of frequency sweep steps.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<Steps> Number of steps
Range: 1 to 10000
*RST: 10

Example: `*RST; :CALC:FORM GDEL`
Select the group delay calculation for the active trace.
`CALC:GDAP:SCO 15`
Select an aperture of 15 steps.

Manual operation: See "Aperture Points" on page 443

CALCulate<Chn>:IAVerage:MODE <Mode>

Selects the quantities to be averaged if infinite averaging is enabled (`CALCulate<Chn>:IAVerage[:STATe] ON`.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<Mode> MAGPhase | REIMag
Selects the averaging mode.

MAGPhase
Averaging of magnitude and phase (default) of the complex trace value

REIMag
Averaging of real and imaginary parts of the complex trace value

Manual operation: See "Mode" on page 475

CALCulate<Chn>:IAVerage[:STATe] <Boolean>

Turns infinite averaging on or off.

The averaging mode can be selected using `CALCulate<Chn>:IAVerage:MODE`.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<Boolean> ON | OFF | 1 | 0

*RST: 0

Manual operation: See "Infinite Average/Restart" on page 475

CALCulate<Chn>:PHOLd <HoldFunc>

Enables, disables, or restarts the max hold and the min hold functions.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<HoldFunc> MIN | MAX | OFF

MIN - Enable the min hold function.
MAX - enable the max hold function.
OFF - disable the max hold or min hold function.

*RST: OFF

Example:
```
*RST; :CALC:PHOL MAX
```
Reset the instrument and enable the max hold function.
```
CALC:PHOL OFF; PHOL MAX
```
Restart max hold.

Manual operation: See "Hold" on page 473

CALCulate:PIPelining:DATA? [<DutId>]

Queries the availability of channel results (to be queried, e.g., using `CALCulate<Chn>:DATA`) in *single-sweep* mode.

The string result `'<true|false>,<channel nos>'` indicates:

- `false`, if the sweep is still running
- `true`, if the sweep has been completed

- The comma-separated list of `channels nos` for which new measurement data are available (since the last call of `CALC:PIP:DATA?`)

Channels measured in *continuous* mode are not considered, and not returned.

Note:

- Because this query blocks the parser thread and the GUI thread until measurement data is available, a `timeout error -365` is returned after 10 seconds wait time.
- If no more data are available, the query returns an `execution error -200`

Query parameters:

<DutId> Not supported yet

Example:

```
// Set up measurements on channels 1 to 4
// Channels 1, 3, and 4:
// - single-sweep mode (INIT<Ch>:CONT OFF)
// - sweep count 1 (SWE:COUN:ALL 1)
// - alternated driving mode (SENS<Ch>:COUP NONE)
// Channel 2: continuous mode (INIT<Ch>:CONT ON).
INIT:PIPe:STAT ON //enable pipelining
INIT:IMM:ALL // start a new single sweep sequence
CALC:PIP:DATA?    // assume a query result 'false, 1'
// --> sweep still running, data available for channel 1
// Query data using CALC1:DATA?
CALC:PIP:DATA?    // assume a query result 'true, 3,4'
// --> sweep completed, data available for channels 3 and 4
// Query data using CALC3:DATA? and CALC4:DATA?
CALC:PIP:DATA?
-200,"Execution error;:CALCulate1:PIPelining:DATA?"
```

Usage: Query only

Manual operation: See "Pipelining" on page 561

CALCulate<Chn>:SMOothing:APERture <SmoothAperture>

Defines how many measurement points are averaged to smooth the trace.

Suffix:

<Chn> Channel number used to identify the active trace

Parameters:

<SmoothAperture> Smoothing aperture. A smoothing aperture of n % means that the smoothing interval for each sweep point i with stimulus value x_i is equal to [x_i - span*n/200, x_i + span*n/200], and that the result of i is replaced by the arithmetic mean value of all measurement points in this interval.

Range: 0.05% to 100%.
*RST: 1
Default unit: %

Example: `*RST; :CALC:SMO ON`
Activate smoothing for the default trace.
`CALC:SMO:APER 0.5`
Reduce the smoothing aperture to 0.5 %.

Manual operation: See "Aperture" on page 472

CALCulate<Chn>:SMOothing[:STATe] <Boolean>

Enables or disables smoothing for trace no. `<Chn>`.

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<Boolean> ON | OFF - smoothing on or off.
*RST: OFF

Example: See `CALCulate<Chn>:SMOothing:APERture`

Manual operation: See "Smoothing" on page 472

CALCulate<Chn>:TRACking:MODE <TrackingMode>

Enables or disables tracking, i.e. shifting the active trace of channel <Chn> relative to the formatted trace value at a configurable reference point (see `CALCulate<Chn>:TRACking:SWEep:POINt` on page 1174).

Suffix:
<Chn>

Parameters:
<TrackingMode> OFF | AMPLitude | PHASe | REAL | IMAGinary | COMPlex
Trace value format for the shift (see manual operation).

Manual operation: See "Tracking Mode/Ref Sweep Point" on page 474

CALCulate<Chn>:TRACking:SWEep:POINt <RefSweepPoint>

Defines the reference point for tracking, if enabled for the active trace of channel <Chn> (see `CALCulate<Chn>:TRACking:MODE`).

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<RefSweepPoint> Sweep point number of the reference point

Manual operation: See "Tracking Mode/Ref Sweep Point" on page 474

CALCulate:TDVSwr[:STATe] <Boolean>

Enables/disables time domain site VSWR measurements.

Parameters:

<Boolean> ON | OFF – TD-VSWR on or off.
*RST: OFF

Manual operation: See "TD-VSWR" on page 360

7.3.2 CONFigure commands

The `CONFigure...` commands create and delete channels or traces, assign channel and trace names, and define the channel sweep order. The commands are device-specific.

CONFigure:CHANnel:CATalog? 1175
CONFigure:CHANnel:MEASure:ACTive[:STATe] 1176
CONFigure:CHANnel:MEASure:ALL[:STATe] 1176
CONFigure:CHANnel:MEASure:OPTimized 1176
CONFigure:CHANnel<Ch>:MEASure[:STATe] 1177
CONFigure:CHANnel<Ch>:NAME 1178
CONFigure:CHANnel<Ch>:NAME:ID? 1178
CONFigure:CHANnel<Ch>[:STATe] 1178
CONFigure:CHANnel<Ch>:TRACe:CATalog? 1179
CONFigure:CHANnel<Ch>:TRACe:REName 1179
CONFigure:TRACe:CATalog? 1180
CONFigure:TRACe<Trc>:CHANnel:NAME? 1180
CONFigure:TRACe<Trc>:CHANnel:NAME:ID? 1181
CONFigure:TRACe:MEM:ACTive 1181
CONFigure:TRACe<Trc>:NAME 1181
CONFigure:TRACe<Trc>:NAME:ID? 1182
CONFigure:TRACe<Trc>:REName 1182
CONFigure:TRACe:WINDow? 1182
CONFigure:TRACe:WINDow:TRACe? 1183

CONFigure:CHANnel:CATalog?

Returns the numbers and names of all channels in the current recall set. The response is a string containing a comma-separated list of channel numbers and names; see example below. If all channels have been deleted the response is an empty string ("").

Example:

```
*RST; :CONF:CHAN2:STAT ON; NAME 'New Channel'
```

Create channel 2 and assign the channel name "New Channel".

```
CONF:CHAN:CAT?
```

Query all channels and their names. As a default channel no. 1 is created on `*RST`, the response is `'1,Ch1,2,New_Channel'`.

```
CONF:CHAN:NAME:ID? 'New Channel'
```

Query the channel number for the channel named "New Channel". The response is `2`.

Usage: Query only

Manual operation: See "Channel table" on page 673

CONFigure:CHANnel:MEASure:ACTive[:STATe] <Boolean>

If set to ON (1), only the active channel is measured. All other channels are switched off temporarily, i.e. until `CONFigure:CHANnel:MEASure:ACTive` is set to `OFF`.

Parameters:
<Boolean> `ON (1) | OFF (0)`

Manual operation: See "Only Active Channel On" on page 672

CONFigure:CHANnel:MEASure:ALL[:STATe] <Boolean>

Enables or disables the sweep in all channels of the active recall set. This command can be used in combination with `CONFigure:CHANnel<Ch>:MEASure[:STATe]` to optimize the measurement speed.

Parameters:
<Boolean> ON | OFF
*RST: ON

Example: See `CONFigure:CHANnel<Ch>:MEASure[:STATe]`

Manual operation: See "Continuous / Single" on page 559

CONFigure:CHANnel:MEASure:OPTimized <Order>[, <Ch1>, <Ch2>, ...]

This setting tells the firmware to optimize the channel switching times for a particular channel sweep order.

Note that the optimization can only be applied if **all** channels in the current setup are in single sweep mode (`INITiate<Ch>:CONTinuous OFF`, and if there is no ongoing sweep sequence. Any change in the current setup or a wrong sweep sequence lets the analyzer fall back to unoptimized channel switching.

Setting parameters:
<Order> AUTO | MANual

AUTO
Optimize the switching times for the channel order that is automatically used with `INIT:ALL` (increasing order of channel numbers).

MANual
Optimize the switching times for the channel order manually specified using channel numbers <Ch1>, <Ch2>, ...

<Ch1>, <Ch2>, ... Can only and must be specified for MANual optimized channel ordering. Defines the order in which the channels are measured. To get the optimization to work, you have to sweep the channels manually, and in the specified order: `INIT<Ch1>`, `INIT<Ch2>`, ...

Example: `*RST;`
`*RST` creates channel 1, trace 1, and diagram 1.
`:INITiate1:CONTinuous OFF`
enable single sweep mode
```
CALC2:PAR:SDEF 'Ch2Tr1', 'S11'
CALC3:PAR:SDEF 'Ch3Tr1', 'S11'
CALC4:PAR:SDEF 'Ch4Tr1', 'S11'
```
create additional channels 2, 3 and 4 with one trace each
```
DISP:WIND1:TRAC2:FEED 'CH2TR1'
DISP:WIND1:TRAC3:FEED 'CH3TR1'
DISP:WIND1:TRAC4:FEED 'CH4TR1'
```
assign these traces to diagram 1
`:CONF:CHAN:MEAS:OPT AUTO; :INITiate1:IMMediate:ALL`
optimize the switching times for the AUTOmatic channel order 1,2,3,4, and restart the sweep in all channels
`:CONF:CHAN:MEAS:OPT MAN, 2,3,1,4; :INIT2; :INIT3; :INIT1; :INIT4`
optimize the switching times for channel order 2,3,1,4 and restart the sweeps in the same order

Usage: Setting only

CONFigure:CHANnel<Ch>:MEASure[:STATe] <Boolean>

Enables or disables the sweep in channel no. `<Ch>`. This command can be used to restrict the measurement in a subset of channels in order to optimize the measurement speed.

Suffix:
<Ch> Number of an existing channel.

Parameters:
<Boolean> ON | OFF
*RST: ON (all existing channels)

Example: `*RST; :CONFigure:CHANnel2 ON; CHANnel3 ON`
Create channels 2 and 3, in addition to the default channel no. 1. The analyzer performs sweeps in all three channels.
`CONFigure:CHANnel:MEASure:ALL OFF`
Disable the measurement in all channels
`CONFigure:CHANnel2:MEASure ON`
(Re-)enable the measurement in channel no. 2. The analyzer measures in channel 2; the channels no. 1 and 3 are not measured.

Manual operation: See "Continuous / Single" on page 559

CONFigure:CHANnel<Ch>:NAME <ChannelName>

Assigns a name to channel number <Ch>. The channel must be created before (CONFigure:CHANnel<Ch>[:STATe] ON). Moreover it is not possible to assign the same name to two different channels. CONFigure:CHANnel:CATalog? returns a list of all defined channels with their names.

Suffix:
<Ch> Number of an existing channel.

Parameters:
<ChannelName> Channel name, e.g. 'Channel 4'.
*RST: 'Ch1'

Example: See CONFigure:CHANnel:CATalog?

Manual operation: See "Channel table" on page 673

CONFigure:CHANnel<Ch>:NAME:ID? <ChannelName>

Queries the channel number (numeric suffix) of a channel with known channel name. A channel name must be assigned before (CONFigure:CHANnel<Ch>:NAME <ChannelName>). CONFigure:CHANnel:CATalog? returns a list of all defined channels with their names.

Suffix:
<Ch> Channel number. This suffix is not relevant and may be omitted (the command returns the actual channel number).

Query parameters:
<ChannelName> Channel name, e.g. 'Channel 4'.

Example: See CONFigure:CHANnel:CATalog?

Usage: Query only

Manual operation: See "Table Area" on page 453

CONFigure:CHANnel<Ch>[:STATe] <Boolean>

Creates or deletes channels or queries for their existence.

Note that CONFigure:CHANnel<Ch> ON creates channels based on the settings of the **active channel** (if any). Use CALCulate:PARameter:NEW:CHANnel to create a channel with default settings.

Suffix:
<Ch> Channel number to be created, deleted or queried

Parameters:
<Boolean>

ON
In set direction, creates channel no. <Ch> if it does not already exist, and/or sets it as the active channel.
The new channel does not contain traces, so that no measurement can be initiated. Use:
– CALCulate<Ch>:PARameter:COPY:CHANnel [OFF] to copy a channel including its traces
– CALCulate<Ch>:PARameter:SDEFine <TraceName>, <Result> to create traces (or traces and channels in one go).

OFF
Deletes channel no. <Ch>, if it exists. Otherwise an error is returned.
In contrast to manual control, where one channel and trace always remain, in remote control it is possible to remove all channels.

*RST: ON for channel no. 1 (created on *RST), OFF for all other channels.

Example: See CONFigure:CHANnel:CATalog?

Manual operation: See "Delete Channel" on page 672

CONFigure:CHANnel<Ch>:TRACe:CATalog?

Returns the numbers and names of all traces in channel no. <Ch>. The response is a string containing a comma-separated list of trace numbers and names; see example. If all traces have been deleted the response is an empty string ("").

Tip: Use CONFigure:TRACe:CATalog? to query the traces in all channels of the active recall set.

Suffix:
<Ch> Channel number

Example: See CONFigure:TRACe:CATalog?

Usage: Query only

Manual operation: See "Table Area" on page 453

CONFigure:CHANnel<Ch>:TRACe:REName <TraceName>

Assigns a (new) name to the active trace in channel <Ch>.

Suffix:
<Ch> Channel number

Setting parameters:
<TraceName> Trace name, e.g. 'Trace 4'.

Example: `*RST; :CONF:CHAN:TRAC:REN 'Testtrace_1'`
Reset the analyzer to create a default trace in channel 1 and set this trace as the active trace. Rename the trace 'Testtrace_1'.
`CALC:PAR:SDEF 'Testtrace_2', 'S11'`
Create a new trace which will become the active trace in channel no. 1.
`CONF:TRAC:REN 'Testtrace_1', 'Testtrace_3'`
Rename the first trace (which is currently not active) 'Testtrace_3'.

Usage: Setting only

Manual operation: See "Table Area" on page 453

CONFigure:TRACe:CATalog?

Returns the numbers and names of all traces in the current recall set. The response is a string containing a comma-separated list of trace numbers and names, see example below. If all traces have been deleted the response is an empty string ("").

Tip: Use `CONFigure:CHANnel<Ch>:TRACe:CATalog?` to query the traces in a particular channel; see example.

Example: `*RST; :CALC2:PAR:SDEF 'Ch2Trc2', 'S11'`
Create channel 2 and a new trace named Ch2Trc2.
`CONF:TRAC:CAT?`
Query all traces and their names. As a default trace no. 1 is created upon *RST, the response is `'1,Trc1,2,Ch2Trc2'`.
`CONF:CHAN1:TRAC:CAT?`
Query the channels in channel no. 1. The response is `'1,Trc1'`.
`CONF:TRAC:NAME:ID? 'Ch2Trc2'`
Query the trace number for the trace named "Ch2Trc2". The response is `2`.
`CONF:TRAC2:NAME?`
Query the trace name for trace no. 2. The response is `'Ch2Trc2'`.
`CONF:TRAC:CHAN:NAME? 'Ch2Trc2'`
Query the channel name for trace Ch2Trc2. The response is `'Ch2'`.
`CONF:TRAC:CHAN:NAME:ID? 'Ch2Trc2'`
Query the channel number for trace Ch2Trc2. The response is `2`.

Usage: Query only

Manual operation: See "Table Area" on page 453

CONFigure:TRACe<Trc>:CHANnel:NAME? <TraceName>

Queries the channel name for an existing trace named `'<TraceName>'`.

Suffix:
<Trc> Trace number. This suffix is ignored; the trace is referenced by its name.

Query parameters:
<TraceName> Trace name, e.g. `'Ch2Trc2'`.

Example: See `CONFigure:TRACe:CATalog?`

Usage: Query only

CONFigure:TRACe<Trc>:CHANnel:NAME:ID? <TraceName>

Queries the channel number (numeric suffix) for an existing trace named `'<TraceName>'`.

Suffix:
<Trc> Trace number. This suffix is ignored; the trace is referenced by its name.

Query parameters:
<TraceName> Trace name, e.g. `'Ch2Trc2'`.

Example: See `CONFigure:TRACe:CATalog?`

Usage: Query only

CONFigure:TRACe:MEM:ACTive <TraceName>

Selects memory trace <TraceName> as the active memory trace of its data trace.

Parameters:
<TraceName> Name of a memory trace

Manual operation: See "Active Memory" on page 451

CONFigure:TRACe<Trc>:NAME <TraceName>

Assigns a name to an existing trace number `<Trc>`. Note that it is not possible to assign the same name to two different traces. `CONFigure:TRACe:CATalog?` returns a list of all traces in the active recall set with their names.

Suffix:
<Trc> Number of an existing trace.

Parameters:
<TraceName> Trace name, e.g. `'Ch2Trc2'`.
*RST: 'Trc1'

Example: See `CONFigure:TRACe:CATalog?`

CONFigure:TRACe<Trc>:NAME:ID? <TraceName>

Queries the trace number (numeric suffix) of a trace with known trace name. `CONFigure:TRACe:CATalog?` returns a list of all traces in the active recall set with their names.

Suffix:
<Trc> Trace number. This suffix is not relevant and may be omitted (the command returns the actual trace number).

Query parameters:
<TraceName> Trace name, e.g. 'Ch2Trc2'.

Example: See `CONFigure:TRACe:CATalog?`

Usage: Query only

CONFigure:TRACe<Trc>:REName <OldTraceName>, <NewTraceName>

Assigns a new name to a trace. The trace does not have to be the active trace.

Suffix:
<Trc> Trace number. This suffix is ignored; the trace is identified via its `<TraceName>`

Setting parameters:
<OldTraceName> String parameter with old trace name, e.g. `'Trc1'`

<NewTraceName> String parameter with new trace name, e.g. `'S11 Trace'`

*RST: n/a

Example: See `CONFigure:CHANnel<Ch>:TRACe:REName`

Usage: Setting only

Manual operation: See "Table Area" on page 453

CONFigure:TRACe:WINDow? <TraceName>

Returns the trace number within a diagram which is assigned to the trace `<TraceName>` is assigned to. A zero is returned when the trace is not assigned/ displayed.

The trace number is equal to the `<WndTr>` suffix in `DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:FEED` and similiar commands; see example.

Query parameters:
<TraceName> Trace name (string), e.g. `'Trc1'`

Example: See `CONFigure:TRACe:WINDow:TRACe?`

Usage: Query only

CONFigure:TRACe:WINDow:TRACe? <TraceName>

Returns the number of the diagram which the trace `<TraceName>` is assigned to. A zero is returned when the trace is not assigned/displayed.

The diagram number is equal to the `<Wnd>` suffix in `DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:FEED` and similar commands; see example.

Query parameters:

<TraceName> Trace name (string), e.g. `'Trc1'`

Example:

```
*RST; :CALC:PAR:SDEF 'Trc2', 'S11'
```
Create a trace named Trc2.
```
CONF:TRAC:WIND:TRAC? 'Trc2'
```
Query the diagram number for Trc2. The new trace is not displayed, so the response is `0`.
```
DISP:WIND2:STAT ON
```
Create a diagram no. 2.
```
DISP:WIND2:TRAC3:FEED 'Trc2'
```
Display the trace in the new diagram no. 2, assigning the trace number 3.
```
CONF:TRAC:WIND? 'Trc2'
```
Query the diagram number for Trc2. The response is `2`.
```
CONF:TRAC:WIND:TRAC? 'Trc2'
```
Query the trace number for Trc2. The response is `3`.

Usage: Query only

Manual operation: See "Table Area" on page 453

7.3.3 CONTrol commands

The `Control...` commands allow you to configure the following interfaces:

- USER PORT
- Handler I/O connector (Universal Interface, option R&S ZN-B14).
- RFFE GPIO interface (option R&S ZN-B15)

CONTrol:AUXiliary:C[:DATA] 1185
CONTrol<Ch>:GPIO<Port>:RANGe 1186
CONTrol:GPIO<Port>:SENSe:CURRent? 1187
CONTrol:GPIO:SENSe:SUMCurrent? 1187
CONTrol<Ch>:GPIO:SENSe:TRIGger 1187
CONTrol:GPIO<Port>:SENSe:VOLTage? 1188
CONTrol<Ch>:GPIO<Port>:SHUNt? 1188
CONTrol<Ch>:GPIO<Port>[:STATe] 1189
CONTrol<Ch>:GPIO:TIME 1190
CONTrol<Ch>:GPIO<Port>:VOLTage[:DEFault] 1190
CONTrol<Ch>:GPIO:VOLTage:OUTPut 1190
CONTrol:HANDler:A[:DATA] 1191
CONTrol:HANDler:B[:DATA] 1191
CONTrol:HANDler:C[:DATA] 1191

CONTrol:HANDler:D[:DATA] 1191
CONTrol:HANDler:E[:DATA] 1191
CONTrol:HANDler:F[:DATA] 1191
CONTrol:HANDler:G[:DATA] 1191
CONTrol:HANDler:H[:DATA] 1191
CONTrol:HANDler:A:MODE 1191
CONTrol:HANDler:B:MODE 1191
CONTrol:HANDler:C:MODE 1192
CONTrol:HANDler:D:MODE 1192
CONTrol:HANDler[:EXTension]:INDex:STATe 1192
CONTrol:HANDler[:EXTension]:RTRigger:STATe 1192
CONTrol:HANDler:INPut? 1193
CONTrol:HANDler:LOGic 1193
CONTrol:HANDler:OUTPut<Pt>[:DATA] 1193
CONTrol:HANDler:OUTPut<Pt>:USER 1194
CONTrol:HANDler:PASSfail:LOGic 1194
CONTrol:HANDler:PASSfail:MODE 1195
CONTrol:HANDler:PASSfail:POLicy 1196
CONTrol:HANDler:PASSfail:SCOPe 1196
CONTrol:HANDler:PASSfail:STATus? 1196
CONTrol:HANDler:RESet 1197
CONTrol:HANDler:SWEepend 1197
CONTrol<Ch>:RFFE<Bus>:COMMand:DATA 1197
CONTrol<Ch>:RFFE<Bus>:COMMand:SEND 1198
CONTrol<Ch>:RFFE<Bus>:COMMand:SEND? 1198
CONTrol<Ch>:RFFE<Bus>:SETTings:FREQuency 1198
CONTrol<Ch>:RFFE<Bus>:SETTings[:STATe] 1199
CONTrol<Ch>:RFFE<Bus>:SETTings:VOLTage:HIGH 1199
CONTrol<Ch>:RFFE<Bus>:SETTings:VOLTage:IO 1199
CONTrol<Ch>:RFFE<Bus>:SETTings:VOLTage:LOW 1199
CONTrol<Ch>:RFFE<Bus>:TEST:CLOCk 1200
CONTrol<Ch>:RFFE<Bus>:TEST:DATA 1200
CONTrol<Ch>:RFFE<Bus>:TEST:VIO 1200
CONTrol:RFFE<Bus>:TEST:CLOCk:CURRent? 1200
CONTrol:RFFE<Bus>:TEST:DATA:CURRent? 1200
CONTrol:RFFE<Bus>:TEST:VIO:CURRent? 1200
CONTrol<Ch>:RFFE<Bus>:TEST:CLOCk:RANGe 1201
CONTrol<Ch>:RFFE<Bus>:TEST:DATA:RANGe 1201
CONTrol<Ch>:RFFE<Bus>:TEST:VIO:RANGe 1201
CONTrol<Ch>:RFFE<Bus>:TEST:CLOCk:SHUNt? 1201
CONTrol<Ch>:RFFE<Bus>:TEST:DATA:SHUNt? 1201
CONTrol<Ch>:RFFE<Bus>:TEST:VIO:SHUNt? 1201
CONTrol:RFFE<Bus>:TEST:CLOCk:VOLTage? 1202
CONTrol:RFFE<Bus>:TEST:DATA:VOLTage? 1202
CONTrol:RFFE<Bus>:TEST:VIO:VOLTage? 1202
CONTrol<Ch>:RFFE:TEST:OUTPut 1202
CONTrol<Ch>:RFFE:TEST:SENSe:TRIGger 1202
CONTrol<Ch>:RFFE:TEST:TIME 1203
CONTrol<Ch>:SEGMent<SegNr>:SEQuence<SeqNr>:DELay 1203
CONTrol<Ch>:SEGMent<SegNr>:SEQuence<SeqNr>:GPIO<Port>:VOLTage 1204

CONTrol<Ch>:SEGMent<SegNr>:SEQuence<SeqNr>:RFFE<Bus>:COMMand:DATA 1204
CONTrol<Ch>:SEGMent<SegNr>:SEQuence:CLEar:ALL 1205
CONTrol<Ch>:SEGMent<SegNr>:SEQuence:COUNt? 1205
CONTrol<Ch>:SEQuence:CLEar:ALL 1205
CONTrol<Ch>:SEQuence:COUNt? 1206
CONTrol<Ch>:SEQuence<SeqNr>:DELay 1206
CONTrol<Ch>:SEQuence<SeqNr>:GPIO<Port>:VOLTage 1206
CONTrol<Ch>:SEQuence<SeqNr>:RFFE<Bus>:COMMand:DATA 1207

CONTrol:AUXiliary:C[:DATA] <DecValue>

Sets or queries a channel-dependent eight-bit decimal value to control eight independent output signals at the USER PORT connector (lines 8, 9, 10, 11 and lines 16, 17, 18, 19). The output signals are 3.3 V TTL signals which can be used to differentiate between up to 255 independent analyzer states. `CONTrol:AUXiliary:C[:DATA]` itself does not change the analyzer state.

Channel bit definition and activation

The channel bits have the following properties:

- After a `*RST` of the analyzer all channel bits (including the value for the active, sweeping channel no. 1) are set to zero; no signal is applied to pins 8 to 11 and 16 to 19 of the USER PORT connector.
- The value defined with `CONTrol:AUXiliary:C[:DATA]` is assigned to the **active** channel (`INSTrument:NSELect <Ch>`).
- The signals at the USER PORT connector reflect the channel bits of the **measuring** channel, i.e. the channel for which the analyzer performs a sweep. This channel is not necessarily identical with the active channel.
- The signals are switched on as soon as a measurement (sweep) in a channel with non-zero channel bits is started. They are changed whenever a channel with different channel bits becomes the measuring channel.
- The signals at the USER PORT connector are maintained after the analyzer enters the hold state. This happens if all channels use single sweep mode and if all sweep sequences have been terminated.
- Pins 16 to 19 may be reserved for monitoring the drive ports 1 to 4 of the analyzer (`OUTPut:UPORt:ECBits OFF`). This leaves up to 16 different monitored channel states.

Tip: A simple application consists of selecting the channel numbers as parameters for `CONTrol:AUXiliary:C[:DATA]` and monitor the activity of up to 255 different channels at the USER PORT connector; see example below. You can also use the USER PORT output signals as channel-dependent trigger signals for external devices. Use `OUTPut<Ch>:UPORt[:VALue]` to transfer the eight bit value for an arbitrary channel <Ch> in binary representation.

Parameters:

<DecValue> Decimal value. The values correspond to the following states of the USER PORT connector:
0 - no signal at any of the no signal at any of the eight pins 8, 9, 10, 11, 16, 17, 18, 19

1 - output signal at pin 8
2 - output signal at pin 9
3 - output signal at pins 8 and 9
...
255 - output signal at pins 8, 9, 10, 11, 16, 17, 18, 19

Range: 0 to 255
*RST: 0 (no signal)

Example:
```
*RST; :CONT:AUX:C 1
```
Assign the channel bit value 1 to the active channel no. 1. The analyzer performs a measurement in channel no. 1, therefore the output signal at pin 8 is switched on.
```
CONF:CHAN2:STAT ON; :CONT:AUX:C 2
```
Create channel no. 2, causing it to become the active channel, and assign the channel bit value 2. The analyzer performs no measurement in channel no. 2, therefore the output signal is not changed.
```
CALC2:PAR:SDEF 'Ch2Tr1', 'S11'
```
Create a trace named 'Ch2Tr1' and assign it to channel 2. While the analyzer measures in channel 2, the output signal changes from pin 8 to pin 9.

Manual operation: See "Optional Columns" on page 550

CONTrol<Ch>:GPIO<Port>:RANGe <Current Range>

Defines an upper bound of the current to be measured on the respective GPIO pin. The analyzer firmware automatically selects a suitable shunt resistance, which can be queried using `CONTrol<Ch>:GPIO<Port>:SHUNt?`.

Suffix:
<Ch> Channel number

<Port> GPIO port number 1, ..., 10

Parameters:
<Current Range> **Ports 1 to 8:** { $2 \cdot 10^n$ µA | n=1,...,5 }
Ports 9 and 10: {0 mA, 100 mA}
Note: The high resistance configuration of pins 9 and 10 requires FPGA version 6.1.0 or higher. For older versions of the R&S ZN-B15/Z15 Var. 03, pins 9 and 10 have the same current range as pins 1 to 8.

Default unit: A

Example:
```
:CONTrol:GPIO1:RANGe 2uA
:CONTrol:GPIO2:RANGe 20uA
:CONTrol:GPIO3:RANGe 200uA
:CONTrol:GPIO4:RANGe 2mA
:CONTrol:GPIO5:RANGe 20mA
:CONTrol:GPIO9:RANGe 0
:CONTrol:GPIO9:RANGe 100mA
```

Options: R&S ZN-B15/Z15 Var. 03

Manual operation: See "Range / Shunt" on page 682

CONTrol:GPIO<Port>:SENSe:CURRent? [<ALL>]

Returns the results of the current measurement on the related GPIO pin or ALL GPIO pins.

Suffix:

<Port> 1, ..., 10
GPIO port number
If ALL currents are queried, this suffix is ignored.

Query parameters:

<ALL> ALL
Use `ALL` to measure the currents at all GPIO pins.

Return values:

<Results> Measured currents, either a single value or a comma-separated list.
Default unit: A

Usage: Query only

Options: R&S ZN-B15/Z15 Var. 03

Manual operation: See "Voltage, Current" on page 683

CONTrol:GPIO:SENSe:SUMCurrent? <GpioPorts>

Returns the sum of the currents measured at the specified GPIO ports

Query parameters:

<GpioPorts> Comma-separated list of GPIO port numbers
The list must contain minimum two and maximum 10 numbers between 1 and 10 in arbitrary order.

Example: `CONTrol:GPIO:SENSe:SUMCurrent? 1,2,3,4,5`
returns the sum of the currents measured in GPIO ports 1 to 5

Usage: Query only

Options: R&S ZN-B15/Z15 Var. 03

Manual operation: See "Voltage, Current" on page 683

CONTrol<Ch>:GPIO:SENSe:TRIGger

Starts the voltage/current measurements on all GPIO pins.

The measurement time can be defined using `CONTrol<Ch>:GPIO:TIME`.

Suffix:
<Ch> Channel number
This suffix is ignored and can be omitted.

Usage: Event

Options: R&S ZN-B15/Z15 Var. 03

Manual operation: See "Meas" on page 682

CONTrol:GPIO<Port>:SENSe:VOLTage? [<ALL>]

Returns the results of the voltage measurement on the related GPIO pin or ALL GPIO pins.

Suffix:
<Port> GPIO port number.
If ALL voltages are queried, this suffix is ignored and can be omitted.

Query parameters:
<ALL> ALL
Use `ALL` to measure the voltages at all GPIO pins.

Return values:
<Results> Measured voltages, either a single value or a comma-separated list.
Default unit: V

Usage: Query only

Options: R&S ZN-B15/Z15 Var. 03

Manual operation: See "Voltage, Current" on page 683

CONTrol<Ch>:GPIO<Port>:SHUNt?

Returns the shunt resistance (in Ω) selected by the analyzer firmware for the configured current range (see `CONTrol<Ch>:GPIO<Port>:RANGe`).

The dependency between current range and shunt resistance is displayed in the tables below.

Note: The high resistance configuration of pins 9 and 10 requires FPGA version 6.1.0 or higher. For older versions of the R&S ZN-B15/Z15 Var. 03, pins 9 and 10 have the same current range as pins 1 to 8.

Suffix:
<Ch> Channel number

<Port> GPIO port 1, ..., 10

Example:
```
:CONTrol1:GPIO1:RANGe 20mA
:CONTrol1:GPIO1:SHUNt?
```
Returns 10 [Ω]
```
:CONTrol1:GPIO1:RANGe 2mA
:CONTrol:GPIO1:TEST:CLOCk:SHUNt?
```
Returns 100 [Ω]
```
:CONTrol1:GPIO1:RANGe 200uA
:CONTrol:GPIO1:SHUNt?
```
Returns 1000 [Ω]
```
:CONTrol1:GPIO9:RANGe 0
:CONTrol:GPIO9:SHUNt?
```
Returns 100000000 [Ω]
```
:CONTrol1:GPIO9:RANGe 100mA
:CONTrol:GPIO9:SHUNt?
```
Returns 1 [Ω]

Usage: Query only

Options: R&S ZN-B15/Z15 Var. 03

Manual operation: See "Range / Shunt" on page 682

Table 7-11: Pins 1 to 8

	Range				
	2 µA	**20 µA**	**200 µA**	**2 mA**	**20 mA**
Shunt	100 kΩ	10 kΩ	1 kΩ	100 Ω	10 Ω

Table 7-12: Pins 9 and 10

	Range	
	0 mA	**100 mA**
Shunt	100 MΩ	1 Ω

CONTrol<Ch>:GPIO<Port>[:STATe] <EnableInSequence>

Enables/disables GPIO port <port> in the Sweep Sequencer for channel <Ch> (see `CONTrol<Ch>:SEQuence<SeqNr>:GPIO<Port>:VOLTage` and `CONTrol<Ch>:SEGMent<SegNr>:SEQuence<SeqNr>:GPIO<Port>:VOLTage`).

Suffix:
<Ch> Channel number

<Port> GPIO port number

Parameters:
<EnableInSequence> Enabled state

Manual operation: See "Seq." on page 681

CONTrol<Ch>:GPIO:TIME <MeasTime>

Sets the measurement time for the voltage/current measurements on the RFFE and GPIO pins.

Same functionality as `CONTrol<Ch>:RFFE:TEST:TIME`.

Suffix:
<Ch> Channel number
This suffix is ignored and can be omitted: the measurement time is valid for all channels.

Parameters:
<MeasTime> Measurement (= sampling = averaging) time
Range: 95 µs to 100 ms
*RST: 100 ms
Default unit: s

Options: R&S ZN-B15/Z15 Var. 03

Manual operation: See "Meas" on page 682

CONTrol<Ch>:GPIO<Port>:VOLTage[:DEFault] <Voltage>

Sets the output voltage of the respective GPIO port.

This voltage is applied using `CONTrol<Ch>:GPIO:VOLTage:OUTPut`.

Suffix:
<Ch> Channel number

<Port> GPIO port number

Parameters:
<Voltage> Range: -7 to +15 V
Increment: 5 mV
Default unit: V

Manual operation: See "Voltage / Output Voltage" on page 681

CONTrol<Ch>:GPIO:VOLTage:OUTPut

Applies the current output voltage and current range (shunt) settings to the RFFE/ GPIO pins.

Same function as `CONTrol<Ch>:RFFE:TEST:OUTPut` on page 1202.

Suffix:
<Ch> Channel number

Usage: Event

Manual operation: See "Apply" on page 681

CONTrol:HANDler:A[:DATA] <DecValue>
CONTrol:HANDler:B[:DATA] <DecValue>
CONTrol:HANDler:C[:DATA] <DecValue>
CONTrol:HANDler:D[:DATA] <DecValue>
CONTrol:HANDler:E[:DATA] <DecValue>
CONTrol:HANDler:F[:DATA] <DecValue>
CONTrol:HANDler:G[:DATA] <DecValue>
CONTrol:HANDler:H[:DATA] <DecValue>

The setting commands write data to ports A, B, C, D, E, F, G, H. To write data to a port, the port must be configured as an output port (see example). By default, the port lines have negative logic: A "0" at a pin corresponds to a high signal, a "1" to a low signal. The logic can be changed using `CONTrol:HANDler:LOGic POSitive`. When writing to port G, port C must be configured as an output port. When writing to port H, port C and port D must be configured as output ports (see `CONTrol:HANDler:D:MODE`).

The queries read data from ports A, B, C, D, E, F, G, H. If the port is an output port, the queries return the last value that was written to the port.

Parameters:

<DecValue> Decimal representation fo an n-bit binary value. The ranges are:
Port A: 0 to 255 (pins A7 ... A0)
Port B: 0 to 255 (pins B7 ... B0)
Port C: 0 to 15 (pins C3 ... C0)
Port D: 0 to 15 (pins D3 ... D0)
Port E: 0 to 255 (pins D3 ... D0 C3 ... C0)
Port F: 0 to 65535 (pins B7 ... B0 A7 ... A0)
Port G: 0 to 1048575 (pins C3 ... C0 B7 ... B0 A7 ... A0)
Port H: 0 to 16777215 (pins D3 ... D0 C3 ... C0 B7 ... B0 A7 ... A0)
The parameters `MIN`, `MAX`, `DEF` are not supported.
Note: `*RST` or "Preset" do not change the configuration of the Universal Interface. Use `CONTrol:HANDler:RESet` to restore default values.

*RST: n/a (default: 0 (port A, B, and F); ports C, D, and E are configured as input ports.)

Example:

```
CONT:HAND:A:MODE OUTP
```
Configure port A as an output port.
```
CONT:HAND:A 192
```
Write data to port A.
```
CONT:HAND:B:MODE INP
```
Configure port B as an input port.
```
CONT:HAND:B?
```
Read data from port B.

CONTrol:HANDler:A:MODE <Mode>
CONTrol:HANDler:B:MODE <Mode>

CONTrol:HANDler:C:MODE <Mode>
CONTrol:HANDler:D:MODE <Mode>

Controls the direction of the data flow at ports A, B, C, D. The direction at the combined ports E, F, G, H is according to the configuration at the other ports.

Parameters:

<Mode> INPut | OUTPut

INPut – Input of data at the port
OUTPut – Output of data at the port
Note: `*RST` or "Preset" do not change the configuration of the Universal Interface. Use `CONTrol:HANDler:RESet` to restore default values.

*RST: n/a (default: Port A and B: OUTPut (also valid for port F); port C and D: INPut (also valid for port E). Ports G and H have mixed default modes.)

Example: See `CONTrol:HANDler:A[:DATA]`

CONTrol:HANDler[:EXTension]:INDex:STATe <Boolean>

Selects the digital signal that is routed to pin 20 of the Universal Interface connector.

Parameters:

<Boolean> ON - /INDEX signal at pin 20
OFF - /PORT_B6 signal at pin 20

*RST: n/a (default: OFF)
Note: `*RST` or "Preset" do not change the configuration of the Universal Interface. Use `CONTrol:HANDler:RESet` to restore default values.

Example:
```
CONT:HAND:EXT:IND:STAT ON
```
Route the /INDEX signal to pin 20.
```
CONT:HAND:EXT:RTR:STAT ON
```
Route the /READY_FOR_TRIGGER signal to pin 21.
```
CONT:HAND:RES
```
Restore the default state: Pins no. 20 and 21 are available for port B input/output signals.

CONTrol:HANDler[:EXTension]:RTRigger:STATe <Boolean>

Selects the digital signal that is routed to pin 21 of the Universal Interface connector.

Parameters:

<Boolean> ON - /READY_FOR_TRIGGER
OFF - /PORT_B7 signal at pin 21
Note: `*RST` or "Preset" do not change the configuration of the Universal Interface. Use `CONTrol:HANDler:RESet` to restore default values.

*RST: n/a (default: OFF)

Example: See `CONTrol:HANDler[:EXTension]:INDex:STATe`

CONTrol:HANDler:INPut?

Queries whether a high to low transition occurred at the /INPUT 1 line (pin 2) of the Universal Interface since the last `CONTrol:HANDler:INPut?` query. The query resets the counter to zero.

A negative pulse fed to this line also causes the /OUTPUT 1 and /OUTPUT 2 lines (pins 3 and 4) to change to low.

Return values:

<NumberOfTrans> 0, if no transition was detected since last query
1, if one or more transitions were detected.

Example: `CONTrol:HANDler:INPut?`
Query whether a high to low transition occurred.

Usage: Query only

CONTrol:HANDler:LOGic <Logic>

Selects the logic of the data ports A to H of the Universal Interface. For output ports, a change in logic reverses the state of the output lines immediately. For input ports, a change in logic will be reflected next time when data is read.

Parameters:

<Logic> POSitive | NEGative

POSitive – 0 = low, 1 = high
NEGative – 0 = high, 1 = low
Note: `*RST` or "Preset" do not change the configuration of the Universal Interface. Use `CONTrol:HANDler:RESet` to restore default values.

*RST: n/a (default: NEGative)

Example: `CONTrol:HANDler:LOGic POS`
Change the logic of the data ports to positive.

CONTrol:HANDler:OUTPut<Pt>[:DATA] <BinValue>

Writes a 0 or 1 to the output ports (pin 3 or 4) of the Universal Interface connector. The port lines have negative logic: A "0" corresponds to a high signal, a "1" to a low signal.

The query reads the last value that has been written to the output port.

Suffix:

<Pt> Output port number:
1 - /OUTPUT1 (pin 3)
2 - /OUTPUT2 (pin 4)
The parameters MIN, MAX, DEF are not supported.
Note: *RST or "Preset" do not change the configuration of the Universal Interface. Use CONTrol:HANDler:RESet to restore default values.

Parameters:

<BinValue> 0 - high
1 - low
*RST: n/a (default: 0)

Example: CONT:HAND:OUTP2:DATA 0
Set the /OUTPUT2 line (pin 4) to 0 (current state of /OUTPUT2).
CONT:HAND:OUTP2:USER 1
Define the next state of the /OUTPUT2 line as 1 (low). /OUTPUT2 will go from 0 to 1 when the analyzer receives a negative pulse on the /INPUT1 line (pin 2).

CONTrol:HANDler:OUTPut<Pt>:USER <BinValue>

Defines the state of the output ports (pin 3 or 4) of the Universal Interface connector after the next negative pulse on the /INPUT1 line (pin 2).

Suffix:

<Pt> Output port number:
1 - /OUTPUT1 (pin 3)
2 - /OUTPUT2 (pin 4)
The parameters MIN, MAX, DEF are not supported.
Note: *RST or "Preset" do not change the configuration of the Universal Interface. Use CONTrol:HANDler:RESet to restore default values.

Parameters:

<BinValue> 0 - high 1 - low
*RST: n/a (default: 0)

Example: See CONTrol:HANDler:OUTPut<Pt>[:DATA]

CONTrol:HANDler:PASSfail:LOGic <Logic>

Specifies the the logic of the /PASS FAIL line (pin 33) of the Universal Interface.

Parameters:

<Logic> POSitive | NEGative

POSitive – high (+5V) means PASS, low (0V) means FAIL
NEGative – low means PASS, high means FAIL
Note: `*RST` or "Preset" do not change the configuration of the Universal Interface. Use `CONTrol:HANDler:RESet` to restore default values.

*RST: n/a (default: NEGative)

Example: See `CONTrol:HANDler:PASSfail:MODE`

CONTrol:HANDler:PASSfail:MODE <Mode>

Specifies the default logical pass/fail state and the timing of the /PASS FAIL line (pin 33). The /PASS FAIL STROBE (pin 36) is set after the /PASS FAIL line; see Section 13.3.4.4, "Timing of control signals", on page 1714.

If the mode is PASS or FAIL, the /PASS FAIL line is returned to its default state when the analyzer is ready for a new measurement (/READY FOR TRIGGER).

Parameters:

<Mode> NOWait | PASS | FAIL

NOWait – the /PASS FAIL line is set as soon as a failure condition occurs.
PASS – the line stays in PASS state (as defined by `CONTrol:HANDler:PASSfail:LOGic`) until a sweep end condition (determined by `CONTrol:HANDler:PASSfail:SCOPe`) occurs.
FAIL – the line stays in FAIL state until a sweep end condition occurs.
Note: `*RST` or "Preset" do not change the configuration of the Universal Interface. Use `CONTrol:HANDler:RESet` to restore default values.

*RST: n/a (default: NOWait)

Example: Configure the /SWEEP END (pin 34) and /PASS FAIL (pin 33) signals:

```
CONTrol:HANDler:SWEepend GLOBal
```

Set the /SWEEP END line to low when all sweeps in all channels are complete.

```
CONTrol:HANDler:PASSFail:MODE PASS
```

Set the default state of the /PASS FAIL line to PASS.

```
CONTrol:HANDler:PASSFail:SCOPe GLOBal
```

Set the /PASS FAIL line when all sweeps in all channels are complete.

```
CONTrol:HANDler:PASSFail:LOGic POSitive
```

Set the /PASS FAIL line to positive logic.

```
CONTrol:HANDler:PASSFail:POLicy ALLTests
```

Return pass only if all tests pass.

CONTrol:HANDler:PASSfail:POLicy <Policy>

Specifies how the global pass/fail status (CONTrol:HANDler:PASSfail:STATus? on page 1196) is calculated.

Parameters:

<Policy> ALLTests | ALLMeas

ALLTests – the status is PASS if all limit checks in all measurements (traces) pass.
ALLMeas – the status is PASS if a limit check is defined for all measurements (traces) and all limit checks pass. It is FAIL if one or more traces have no associated limit check, or if at least one limit check fails.
Note: *RST or "Preset" do not change the configuration of the Universal Interface. Use CONTrol:HANDler:RESet to restore default values.

Example: See CONTrol:HANDler:PASSfail:MODE

CONTrol:HANDler:PASSfail:SCOPe <Scope>

Specifies the "sweep end" condition that will cause the /PASS FAIL line (pin 33) to report the status of the global limit check.

Note: This setting is not valid if the pass/fail mode is NOWait (CONTrol:HANDler:PASSfail:MODE NOWait).

Parameters:

<Scope> GLOBal | CHANnel

CHANnel – when all the sweeps for each channel are complete
GLOBal – when all sweeps in all channels are complete
Note: *RST or "Preset" do not change the configuration of the Universal Interface. Use CONTrol:HANDler:RESet to restore default values.

*RST: n/a (default: GLOBal)

Example: See CONTrol:HANDler:PASSfail:MODE

CONTrol:HANDler:PASSfail:STATus?

Returns the global pass/fail status of the last measurement.

Return values:

<Status> PASS | FAIL | NONE

PASS – all measurements that are not in single sweep mode (on hold) have been swept, and all limit checks have been passed.
FAIL – all measurements that are not in single sweep mode (on hold) have been swept, at least one limit check failed according to the specified pass/fail policy (`CONTrol:HANDler:PASSfail:POLicy`).
NONE – no pass/fail status available, e.g. because the measurement is in progress or because no limit check has been defined.

*RST: n/a

Example: Preparations: Configure and enable a limit check. Start a measurement and wait until the sweep is complete.
`CONTrol:HANDler:PASSfail:STATus?`
Query the result of the global limit check.

Usage: Query only

CONTrol:HANDler:RESet

Restores the default states of the `CONTrol:HANDler...` commands including the data port values.

Example: See `CONTrol:HANDler[:EXTension]:INDex:STATe`

Usage: Event

CONTrol:HANDler:SWEepend <SweepEnd>

Specifies the event that will cause the /SWEEP END line (pin 34) to go low; see Section 13.3.4.4, "Timing of control signals", on page 1714.

Parameters:

<SweepEnd> SWEep | CHANnel | GLOBal

SWEep – every time a sweep is complete
CHANnel – when all the sweeps for each channel are complete
GLOBal – when all sweeps in all channels are complete
Note: `*RST` or "Preset" do not change the configuration of the Universal Interface. Use `CONTrol:HANDler:RESet` to restore default values.

*RST: n/a (default: GLOBal)

Example: See `CONTrol:HANDler:PASSfail:MODE`

CONTrol<Ch>:RFFE<Bus>:COMMand:DATA <Command>

Defines an RFFE command for channel <Ch> and RFFE bus interface <Bus>, which can be executed using `CONTrol<Ch>:RFFE<Bus>:COMMand:SEND` (write-only) or `CONTrol<Ch>:RFFE<Bus>:COMMand:SEND? <BytesToRead>` (with read-back).

For details and background information see the "MIPI Alliance Specification for RF Front-End Control Interface".

Suffix:
<Ch> Channel number

<Bus> RFFE bus interface number

Parameters:
<Command> 3, 5, ..., 35, or 37 **hexadecimal digits** (0-F), defining the command to be executed:
- digit 1 is the slave address,
- digits 2 and 3 specify the command number and
- the remaining digits represent the data part with up to 17 bytes (0, 2, ..., 32, or 34 hex digits).

Manual operation: See "RFFE Command" on page 677

CONTrol<Ch>:RFFE<Bus>:COMMand:SEND
CONTrol<Ch>:RFFE<Bus>:COMMand:SEND? <BytesToRead>

Sends the RFFE command for channel `<Ch>` and RFFE bus interface `<Bus>`.

In its "set" form, it is a pure write command. In its query form, the command attempts to read `<BytesToRead>` bytes back: the result is returned in the form `<ParityBit>,<ReadError>,'<result>'`, where

- `<ReadError>`==1 indicates a parity mismatch
- `<result>` consits of 2·`<BytesToRead>` hexadecimal digits.

The command has to be defined previously using `CONTrol<Ch>:RFFE<Bus>:COMMand:DATA`. On a R&S ZN-B15/Z15 **var. 03**, before the command is executed the related shunt resistance is set to its minimum possible value.

For details and background information see the "MIPI Alliance Specification for RF Front-End Control Interface".

Suffix:
<Ch> Chanel number

<Bus> RFFE bus interface

Query parameters:
<BytesToRead> The number of bytes to be read back from the RFFE interface.
*RST: 0 to 16

Manual operation: See "SEND" on page 678

CONTrol<Ch>:RFFE<Bus>:SETTings:FREQuency <ClockFrequency>

Sets/gets the clock frequency for channel <Ch> and RFFE bus <Bus>.

Suffix:
<Ch> Channel number

<Bus> 1 or 2
RFFE bus number

Parameters:
<ClockFrequency> Clock rate.
Possible values are 52/1664 MHz, 52/1663 MHz, ..., 52/2 MHz.
Range: 31.25 kHz to 26 MHz
Default unit: Hz

Manual operation: See "CLK, VIO, VLow, VHigh" on page 677

CONTrol<Ch>:RFFE<Bus>:SETTings[:STATe] <EnableInSequence>

Enables/disables RFFE bus interface <Bus> in the Sweep Sequencer for channel <Ch> (see `CONTrol<Ch>:SEQuence<SeqNr>:RFFE<Bus>:COMMand:DATA` and `CONTrol<Ch>:SEGMent<SegNr>:SEQuence<SeqNr>:RFFE<Bus>:COMMand:DATA`).

Suffix:
<Ch> Channel number

<Bus> RFFE bus interface number

Parameters:
<EnableInSequence> Enabled state

Manual operation: See "Seq." on page 677

CONTrol<Ch>:RFFE<Bus>:SETTings:VOLTage:HIGH <VoltageHigh>
CONTrol<Ch>:RFFE<Bus>:SETTings:VOLTage:IO <IOVoltage>
CONTrol<Ch>:RFFE<Bus>:SETTings:VOLTage:LOW <Voltage>

Sets/gets the IO, Low and High voltages for channel <Ch> and RFFE bus interface <Bus>, respectively.

Note that these voltages are only applied while an RFFE command is executed using `CONTrol<Ch>:RFFE<Bus>:COMMand:SEND` or `CONTrol<Ch>:RFFE<Bus>:COMMand:SEND?`.

Suffix:
<Ch> Channel number

<Bus> RFFE bus number

Parameters:
<Voltage> Range: 0 to 2.5 V
Increment: 0.001 V
Default unit: V

Manual operation: See "CLK, VIO, VLow, VHigh" on page 677

CONTrol<Ch>:RFFE<Bus>:TEST:CLOCk <Test Clock Voltage>
CONTrol<Ch>:RFFE<Bus>:TEST:DATA <Test Data Voltage>
CONTrol<Ch>:RFFE<Bus>:TEST:VIO <OutputVoltage>

These commands define the output voltages for the voltage/current measurements on the RFFE pins.

Note: The voltages for DATA and CLOCk are always identical; their values cannot be set independently.

The output voltages are applied using `CONTrol<Ch>:RFFE:TEST:OUTPut`, the voltage/current measurements are started using `CONTrol<Ch>:RFFE:TEST:SENSe:TRIGger`.

Suffix:
<Ch> Channel number

<Bus> RFFE bus interface 1 or 2

Parameters:
<OutputVoltage>
Range: 0 V to 2.5 V
Increment: 1 mV
*RST: 0 V
Default unit: V

Example:
```
:CONTrol:RFFE1:TEST:DATA 1V
:CONTrol:RFFE1:TEST:CLOC?
```
DATA and CLOCK are set simultaneously, so this should return 1V
```
:CONTrol:RFFE1:TEST:VIO 1V
:CONTrol:RFFE2:TEST:CLOCk 2V
:CONTrol:RFFE2:TEST:DATA?
```
DATA and CLOCK are set simultaneously, so this should return 2V
```
:CONTrol:RFFE2:TEST:VIO 2V
```

Options: R&S ZN-B15/Z15 Var. 03

Manual operation: See "Output Voltage" on page 679

CONTrol:RFFE<Bus>:TEST:CLOCk:CURRent?
CONTrol:RFFE<Bus>:TEST:DATA:CURRent?
CONTrol:RFFE<Bus>:TEST:VIO:CURRent?

Returns the results of the current measurement on the related RFFE pin.

Suffix:
<Bus> RFFE bus interface 1 or 2

Return values:
<Measured> Measured current

Usage: Query only

Options: R&S ZN-B15 Var. 03

Manual operation: See "Voltage, Current" on page 680

CONTrol<Ch>:RFFE<Bus>:TEST:CLOCk:RANGe <Clock Current Range>
CONTrol<Ch>:RFFE<Bus>:TEST:DATA:RANGe <Data Current Range>
CONTrol<Ch>:RFFE<Bus>:TEST:VIO:RANGe <CurrentRange>

Defines the (upper bound of the) current range for the voltage/current measurement on the respective RFFE pin. The analyzer firmware automatically selects a suitable shunt resistance, which can be queried using `CONTrol<Ch>:RFFE<Bus>:TEST:CLOCk|DATA|VIO:SHUNt`.

Suffix:
<Ch> Channel number

<Bus> RFFE bus interface 1 or 2

Parameters:
<CurrentRange> Range: 2 x 10^n µA with n=1,...,5
Default unit: A

Example: See `CONTrol<Ch>:RFFE<Bus>:TEST:CLOCk|DATA|VIO:SHUNt`

Options: R&S ZN-B15/Z15 Var. 03

Manual operation: See "Range / Shunt" on page 679

CONTrol<Ch>:RFFE<Bus>:TEST:CLOCk:SHUNt?
CONTrol<Ch>:RFFE<Bus>:TEST:DATA:SHUNt?
CONTrol<Ch>:RFFE<Bus>:TEST:VIO:SHUNt?

Returns the shunt resistance (in Ω) selected by the analyzer firmware for the configured current range (see `CONTrol<Ch>:RFFE<Bus>:TEST:CLOCk|DATA|VIO:RANGe`).

The dependency between current range and shunt resistance is displayed in the table below.

Suffix:
<Ch> Channel number

<Bus> RFFE bus interface 1 or 2

Example:
```
:CONTrol1:RFFE1:TEST:DATA:RANGe 20.0mA
:CONTrol1:RFFE1:TEST:CLOCk:RANGe 2mA
:CONTrol1:RFFE1:TEST:VIO:RANGe 0.2mA
:CONTrol1:RFFE1:TEST:DATA:SHUNt?
```
Returns 10 [Ω]
```
:CONTrol1:RFFE1:TEST:VIO:RANGe 2mA
:CONTrol:RFFE1:TEST:CLOCk:SHUNt?
```
Returns 100 [Ω]
```
:CONTrol1:RFFE1:TEST:VIO:RANGe 0.2mA
:CONTrol:RFFE1:TEST:VIO:SHUNt?
```
Returns 1000 [Ω]

Usage: Query only

Options: R&S ZN-B15/Z15 Var. 03

Manual operation: See "Range / Shunt" on page 679

	2 µA	20 µA	200 µA	2 mA	20 mA
Shunt	10 Ω	100 Ω	1 kΩ	10 kΩ	100 kΩ

CONTrol:RFFE<Bus>:TEST:CLOCk:VOLTage?
CONTrol:RFFE<Bus>:TEST:DATA:VOLTage?
CONTrol:RFFE<Bus>:TEST:VIO:VOLTage?

Returns the results of the voltage measurement on the related RFFE pin.

Suffix:
<Bus> RFFE bus interface 1 or 2

Return values:
<Measured> Measured voltage

Usage: Query only

Options: R&S ZN-B15 Var. 03

Manual operation: See "Voltage, Current" on page 680

CONTrol<Ch>:RFFE:TEST:OUTPut

Applies the current output voltage and current range (shunt) settings to the RFFE/ GPIO pins.

Same function as `CONTrol<Ch>:GPIO:VOLTage:OUTPut`.

Suffix:
<Ch> Channel number

Usage: Event

Options: R&S ZN-B15/Z15 Var. 03

Manual operation: See "Set" on page 680

CONTrol<Ch>:RFFE:TEST:SENSe:TRIGger

Starts the voltage/current measurements on all RFFE pins.

The measurement time can be defined using `CONTrol<Ch>:RFFE:TEST:TIME`.

Suffix:
<Ch> Channel number
This suffix is ignored and can be omitted.

Usage: Event

Options: R&S ZN-B15/Z15 Var. 03

Manual operation: See "Start Meas" on page 680

CONTrol<Ch>:RFFE:TEST:TIME <MeasTime>

Sets the measurement time for the voltage/current measurements on the RFFE and GPIO pins.

Same functionality as `CONTrol<Ch>:GPIO:TIME`.

Suffix:

<Ch> Channel number
This suffix is ignored and can be omitted: the measurement time is valid for all channels.

Parameters:

<MeasTime> Measurement (= sampling = averaging) time

Range: 95 µs to 100 ms
*RST: 100 ms
Default unit: s

Options: R&S ZN-B15/Z15 Var. 03

Manual operation: See "Start Meas" on page 680

CONTrol<Ch>:SEGMent<SegNr>:SEQuence<SeqNr>:DELay <Delay>

For segmented sweeps this command allows to introduce delays between subsequent sequence steps and between the final sequence step and the sweep (segment) start.

The command/switch sequences are defined using `CONTrol<Ch>:SEGMent<SegNr>:SEQuence<SeqNr>:RFFE<Bus>:COMMand:DATA` and `CONTrol<Ch>:SEGMent<SegNr>:SEQuence<SeqNr>:GPIO<Port>:VOLTage`.

Suffix:

<Ch> Channel number

<SegNr> Segment number

<SeqNr> Sequence number, defining the order in which the commands shall be executed.
For every channel, segment and RFFE interface, the sequence numbers must be consecutive, starting at 1; reusing a sequence number overwrites a previously defined command.

Parameters:

<Delay> Delay time

Default unit: s

Manual operation: See "Wait (Sweep Sequencer Table)" on page 685

CONTrol<Ch>:SEGMent<SegNr>:SEQuence<SeqNr>:GPIO<Port>:VOLTage <Voltage>

For segmented sweeps this command allows to define the GPIO voltage settings to be applied at the start of each segment.

The GPIO ports can be enabled/disabled using `CONTrol<Ch>:GPIO<Port>[:STATe]`. Complementary RFFE commands can be defined using `CONTrol<Ch>:SEGMent<SegNr>:SEQuence<SeqNr>:RFFE<Bus>:COMMand:DATA`.

Suffix:

<Ch> Channel number

<SegNr> Segment number

<SeqNr> Sequence number, defining the order in which the commands shall be executed.
For every channel, segment and RFFE interface, the sequence numbers must be consecutive, starting at 1; reusing a sequence number overwrites a previously defined command.

<Port>

Parameters:

<Voltage> Range: -6 to +12 V
Increment: 0.005 V
Default unit: V

Manual operation: See "GPIO columns (sweep sequencer table)" on page 684

CONTrol<Ch>:SEGMent<SegNr>:SEQuence<SeqNr>:RFFE<Bus>:COMMand:DATA <Command>

For segmented sweeps this command allows to define the RFFE command(s) to be executed at the start of each segment.

Command execution on an RFFE interface can be enabled/disabled using `CONTrol<Ch>:RFFE<Bus>:SETTings[:STATe]`. Complementary GPIO switches can be defined using `CONTrol<Ch>:SEGMent<SegNr>:SEQuence<SeqNr>:GPIO<Port>:VOLTage`.

Suffix:

<Ch> Channel number

<SegNr> Segment number

<SeqNr> Sequence number, defining the order in which the commands shall be executed.
For every channel, segment and RFFE interface, the sequence numbers must be consecutive, starting at 1; reusing a sequence number overwrites a previously defined command.

<Bus> RFFE bus interface number

Parameters:

<Command> 3 to 37 **hexadecimal digits** (0-F), defining the command to be executed: digit 1 is the slave address, digits 2 and 3 specify the command number and the remaining digits represent the data part (up to 17 digit **pairs**).

Manual operation: See "RFFE columns (sweep sequencer table)" on page 684

CONTrol<Ch>:SEGMent<SegNr>:SEQuence:CLEar:ALL

For segmented sweeps this command deletes the command/switch sequence for the respective channel and sweep segment (defined using `CONTrol<Ch>:SEGMent<SegNr>:SEQuence<SeqNr>:RFFE<Bus>:COMMand:DATA` and `CONTrol<Ch>:SEGMent<SegNr>:SEQuence<SeqNr>:GPIO<Port>:VOLTage`).

Suffix:

<Ch> Channel number

<SegNr> Segment number

Usage: Event

Manual operation: See "Range" on page 684

CONTrol<Ch>:SEGMent<SegNr>:SEQuence:COUNt?

For segmented sweeps this command queries the length of the command/switch sequence defined for a particular channel and sweep segment.

The command/switch sequences are defined using `CONTrol<Ch>:SEGMent<SegNr>:SEQuence<SeqNr>:RFFE<Bus>:COMMand:DATA` and `CONTrol<Ch>:SEGMent<SegNr>:SEQuence<SeqNr>:GPIO<Port>:VOLTage`.

Suffix:

<Ch> Channel number

<SegNr> Segment number

Usage: Query only

Manual operation: See "Range" on page 684

CONTrol<Ch>:SEQuence:CLEar:ALL

For unsegmented sweeps this command deletes the command/switch sequence for the respective channel (defined using `CONTrol<Ch>:SEQuence<SeqNr>:RFFE<Bus>:COMMand:DATA` and `CONTrol<Ch>:SEQuence<SeqNr>:GPIO<Port>:VOLTage`).

Suffix:

<Ch> Channel number

Usage: Event

Manual operation: See "Range" on page 684

CONTrol<Ch>:SEQuence:COUNt?

For unsegmented sweeps this command queries the length of the command sequence defined for a particular channel (using `CONTrol<Ch>:SEQuence<SeqNr>:RFFE<Bus>:COMMand:DATA`).

Suffix:
<Ch> Channel number

Usage: Query only

Manual operation: See "Range" on page 684

CONTrol<Ch>:SEQuence<SeqNr>:DELay <Delay>

For unsegmented sweeps this command allows to introduce delays between subsequent sequence steps and between the final sequence step and the sweep (segment) start.

The command/switch sequences are defined using `CONTrol<Ch>:SEQuence<SeqNr>:RFFE<Bus>:COMMand:DATA` and `CONTrol<Ch>:SEQuence<SeqNr>:GPIO<Port>:VOLTage`.

Suffix:
<Ch> Channel number

<SeqNr> Sequence number, defining the order in which the commands shall be executed.
For every channel and RFFE interface, the sequence numbers must be consecutive, starting at 1; reusing a sequence number overwrites a previously defined command.

Parameters:
<Delay> Delay time
Default unit: s

Manual operation: See "Wait (Sweep Sequencer Table)" on page 685

CONTrol<Ch>:SEQuence<SeqNr>:GPIO<Port>:VOLTage <Voltage>

For unsegmented sweeps this command allows to define the GPIO voltage settings to be applied at the start of each sweep.

The GPIO ports can be enabled/disabled using `CONTrol<Ch>:GPIO<Port>[:STATe]`. Complementary RFFE commands can be defined using `CONTrol<Ch>:SEQuence<SeqNr>:RFFE<Bus>:COMMand:DATA`.

Suffix:
<Ch> Channel number

<SeqNr> Sequence number, defining the order in which the switches/ commands shall be executed.
For every channel and GPIO port, the sequence numbers must be consecutive, starting at 1; reusing a sequence number overwrites a previous voltage setting.
Delays between subsequent settings/command and - finally - the start of the respective sweep segment can be introduced using `CONTrol<Ch>:SEQuence<SeqNr>:DELay`.

<Port> GPIO port number

Parameters:

<Voltage> Range: -6 to +12 V
Increment: 0.005 V
Default unit: V

Manual operation: See "GPIO columns (sweep sequencer table)" on page 684

CONTrol<Ch>:SEQuence<SeqNr>:RFFE<Bus>:COMMand:DATA <Command>

For unsegmented sweeps this command allows to define the RFFE command(s) to be executed at the start of each sweep.

Command execution on an RFFE interface can be enabled/disabled using `CONTrol<Ch>:RFFE<Bus>:SETTings[:STATe]`. Complementary GPIO switches can be defined using `CONTrol<Ch>:SEQuence<SeqNr>:GPIO<Port>:VOLTage`.

Suffix:

<Ch> Channel number

<SeqNr> Sequence number, defining the order in which the commands shall be executed.
For every channel and RFFE interface, the sequence numbers must be consecutive, starting at 1; reusing a sequence number overwrites a previously defined command.
Delays between subsequent commands and - finally - the start of the respective sweep can be introduced using `CONTrol<Ch>:SEQuence<SeqNr>:DELay`

<Bus> RFFE bus interface number

Parameters:

<Command> 3 to 37 **hexadecimal digits** (0-F), defining the command to be executed: digit 1 is the slave address, digits 2 and 3 specify the command number and the remaining digits represent the data part (up to 17 digit **pairs**).

Manual operation: See "RFFE columns (sweep sequencer table)" on page 684

7.3.4 DIAGnostic commands

The `DIAGnostic...` commands provide access to service and diagnostic routines used in service, maintenance and repair. In accordance with the SCPI standard, all commands are device-specific.

Service functions are password-protected (`SYSTem:PASSword[:CENable]`) and are intended for Rohde & Schwarz service staff. Refer to the service manual for more information.

DIAGnostic:DEVice:STATe 1208
DIAGnostic:DUMP:SIZE 1208
DIAGnostic:PRODuct:OPTion:INFO? 1209
DIAGnostic:SERVice:HWINfo? 1209
DIAGnostic:SERVice:RFPower 1209
DIAGnostic:SERVice:SFUNction 1210
DIAGnostic:SERVice:SWINfo? 1210

DIAGnostic:DEVice:STATe <Filename>

Generates a system report and writes it to the specified ZIP file. See Section 9.3, "Collecting information for technical support", on page 1691.

Setting parameters:

<Filename> String parameter containing the file name. If no path is specified, the file is stored to the directory `C:\Users\Public\Documents\Rohde-Schwarz\Vna\Report`.

*RST: n/a

Example: `DIAG:DEV:STAT 'report_16032011_1120.zip'`
Generate a report and store it to `C:\Users\Public\Documents\Rohde-Schwarz\Vna\Report\report_16032011_1120.zip`. Use the `MMEMory...` commands to rename, move, or delete the file.

Usage: Setting only

Manual operation: See "Save..." on page 826

DIAGnostic:DUMP:SIZE <DumpSize>

Determines the level of detail and hence the size of the dump files created in case of firmware exceptions.

Parameters:

<DumpSize> NONE | MINI | NORMal | LARGe | FULL

Either disables dump file creation (NONE) or determines the level of detail.

Manual operation: See "Error Dump Type" on page 819

DIAGnostic:PRODuct:OPTion:INFO? <Option>, <Detail>

Queries a property of an installed software option, identified by its name.

Query parameters:

<Option> Option name, e.g. `'ZNB-K2'`.

<Detail> DESCription | TYPE | ACTivation | EXPiration | KEY

You can query for an option's DESCription, its KEY and key TYPE, and its ACTivation and EXPiration date (if applicable).

Example: `DIAGnostic:PRODuct:OPTion:INFO? 'ZNB-K2',DESC`
On a R&S ZNB, this returns `'Time Domain Analysis'` (if installed).

Usage: Query only

Manual operation: See "Software Option Info" on page 829

DIAGnostic:SERVice:HWINfo?

Returns basic hardware information about the instrument.

The result comprises the following items (separated by slashes):

- Instrument type
- Number of test ports on the VNA and connected switch matrices
- Number of physical VNA ports
- Part number
- Serial number
- Device ID
- Specifications document version

Example: `:DIAGnostic:SERVice:HWINfo?`
returns something like
`ZNB8/4/4/1334.3330K44/999999/1325.6601K03-900015-Rt/1.00`

Usage: Query only

Manual operation: See "Instrument Information" on page 829

DIAGnostic:SERVice:RFPower <Boolean>

Turns the internal source power at all ports and the power of all external generators on or off. This command is equivalent to `OUTPut<Ch>[:STATe]` .

Parameters:

<Boolean> ON | OFF - switch the power on or off.

*RST: ON

Example: `DIAG:SERV:RFP OFF`
Turn off the RF source power.

DIAGnostic:SERVice:SFUNction <SFIdentifier>

Calls a service function (mainly for internal use).

- Use `SYSTem:PASSword[:CENable]` to activate the required service level.
- Use the query form to read back the data returned by the service function.

Parameters:

<SFIdentifier> Service function in "dotted textual" (example: 'sw.common.memory_usage') or "dotted decimal" (example: '0.1.18.0') representation.

Example: `DIAGnostic:SERVice:SFUNction? 'sw.common.memory_usage'`
This is an "Info Level" service function, i.e. it is not password-protected. It returns the curent memory usage of the analyzer firmware.

Manual operation: See "Service Function" on page 839

DIAGnostic:SERVice:SWINfo?

Returns basic software information about the instrument.

The result comprises the following items (separated by slashes):

- Service tool version
- Base device image version
- Windows version
- Bios version

Example: `:DIAGnostic:SERVice:SWINfo?`
returns something like
`3.68/4.9/Windows10-21H2-LTSC-2021-x64-Production/R&S VNA W10 4.08`

Usage: Query only

Manual operation: See "Instrument Information" on page 829

7.3.5 DISPlay commands

The `DISPlay...` commands control the selection and presentation of graphical and trace information on the screen.

Trace display

Traces are identified by a string parameter defining the trace name (e.g. `CALCulate<Ch>:PARameter:SELect <TraceName>`). In the `DISPlay...` subsystem, traces are assigned to diagrams (`DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:FEED <TraceName>`). While this assignment is valid, the trace is identified by the numeric suffix `<Wnd>`, and the trace name is not needed.

Units for DISPlay... commands

The `DISPlay...` subsystem contains commands to define particular points in the diagram, e.g. to set the scale or a reference value. Some settings require a numeric value and a physical unit, depending on the related parameter type. The following table lists the physical units accepted by the analyzer.

Parameter type	Physical unit
Power	DBM, DB, DBW, W, MW, UW, NW, PW
Voltage	V, MV, UV, NV, PV, DBV, DBMV, DBUV
Phase	DEG, KDEG, MDEG, UDEG, NDEG, PDEG
Group delay	S, MS, US, NS, PS
Impedance	OHM, GOHM, MOHM, KOHM
Admittance	SIE, MSIE, USIE, NSIE
Inductance	H, MH, UH, NH, PH, FH
Capacitance	F, MF, UF, NF, PF, FF
Dimensionless	UNIT, MUNIT, UUNIT, NUNIT, PUNIT, FUNIT

DISPlay:ANNotation:CHANnel[:STATe] 1212
DISPlay:ANNotation:TRACe[:STATe] 1212
DISPlay:ANNotation:MARKer:STIMulus[:STATe] 1213
DISPlay:ANNotation:MARKer:TRACe:INFO[:STATe] 1213
DISPlay:ANNotation:INFO:POSition 1213
DISPlay:ANNotation:INFO:STATe 1213
DISPlay:CMAP:BWSCheme[:STATe] 1214
DISPlay:CMAP:LSCHeme[:STATe] 1214
DISPlay:CMAP:LSSCheme[:STATe] 1214
DISPlay:CMAP:LIMit:FCOLorize[:STATe] 1214
DISPlay:CMAP:LIMit:FSYMbol[:STATe] 1214
DISPlay:CMAP:LIMit[:STATe] 1214
DISPlay:CMAP:EYE:TCHigh:RGB 1215
DISPlay:CMAP:EYE:TCLow:RGB 1215
DISPlay:CMAP:MARKer[:STATe] 1215
DISPlay:CMAP<DispEl>:RGB 1216
DISPlay:CMAP:TRACe:COLor[:STATe] 1217
DISPlay:CMAP:TRACe:RGB 1218
DISPlay:IWINdow:BFILter[:STATe] 1219
DISPlay:IWINdow:MARKer<Mk>[:STATe] 1219
DISPlay:IWINdow[:STATe] 1219
DISPlay:LAYout 1220
DISPlay:LAYout:APPLy 1220
DISPlay:LAYout:DEFine 1220
DISPlay:LAYout:EXECute 1221
DISPlay:LAYout:GRID 1222
DISPlay:LAYout:JOIN 1222
DISPlay:LAYout:OVERlay 1223

DISPlay:LAYout:SPLit....1223
DISPlay:MENU:KEY:ACTion:CATalog?....1223
DISPlay:MENU:KEY:EXECute....1223
DISPlay:MENU:KEY:SELect....1224
DISPlay:MENU:KEY:TOOL:CATalog?....1224
DISPlay:RFSize....1224
DISPlay[:WINDow<Wnd>]:ANNotation:MARKer:INFO[:STATe]....1224
DISPlay[:WINDow<Wnd>]:CATalog?....1225
DISPlay[:WINDow<Wnd>]:MARKer:CATalog?....1225
DISPlay[:WINDow<Wnd>]:MAXimize....1226
DISPlay[:WINDow<Wnd>]:NAME....1226
DISPlay[:WINDow<Wnd>]:OVERview[:STATe]....1226
DISPlay[:WINDow<Wnd>]:SIZE....1227
DISPlay[:WINDow<Wnd>][:STATe]....1227
DISPlay[:WINDow<Wnd>]:TITLe:DATA....1227
DISPlay[:WINDow<Wnd>]:TITLe[:STATe]....1228
DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:CATalog?....1228
DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:COUPle....1228
DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:COUPle:ALL....1229
DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:DELete....1229
DISPlay[:WINDow<Wnd>]:TRACe:EFEed....1230
DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:FEED....1231
DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:LABel:SHOW....1231
DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:SHOW....1232
DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:X:OFFSet....1232
DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:Y:OFFSet....1233
DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:Y[:SCALe]:AUTO....1234
DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:Y[:SCALe]:BOTTom....1234
DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:Y[:SCALe]:TOP....1234
DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:Y[:SCALe]:CONTinuous....1235
DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:Y[:SCALe]:PDIVision....1236
DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:Y[:SCALe]:RLEVel....1236
DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:Y[:SCALe]:RPOSition....1237
DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:ZOOM:BOTTom....1238
DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:ZOOM:TOP....1238
DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:ZOOM:STARt....1239
DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:ZOOM:STOP....1239
DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:ZOOM[:STATe]....1239
DISPlay[:WINDow<Wnd>]:Y[:SCALe]:AUTO....1240

DISPlay:ANNotation:CHANnel[:STATe] <Boolean>
DISPlay:ANNotation:TRACe[:STATe] <Boolean>

Shows or hides the channel/trace list(s) in all diagrams of the current recall set.

Parameters:

<Boolean> ON | OFF - show or hide information element(s).
*RST: ON

Example: `*RST; :DISP:ANN:TRAC OFF; CHAN ON; FREQ OFF`
Create diagram area no. 1 (with default trace) and hide the trace list. Keep the channel list but hide the swept frequency range.

Manual operation: See "Trace Info" on page 802

DISPlay:ANNotation:MARKer:STIMulus[:STATe] <Boolean>

If enabled, the stimulus information (the frequencies of the markers) is included in the marker info field.

Parameters:
<Boolean> *RST: ON (1)

Manual operation: See "Stimulus Info" on page 524

DISPlay:ANNotation:MARKer:TRACe:INFO[:STATe] <Boolean>

If enabled, the marker info field displays only marker information for the active trace.

Parameters:
<Boolean> *RST: OFF (0)

Manual operation: See "Active Trace Only" on page 524

DISPlay:ANNotation:INFO:POSition <InfoTableMode>

Defines or queries the position of the info table (shown or hidden using `DISPlay:ANNotation:INFO:STATe ON | OFF`).

Parameters:
<InfoTableMode> BOTTom | LEFT | RIGHt
*RST: BOTTom

Manual operation: See "Info Table: Show / Position" on page 803

DISPlay:ANNotation:INFO:STATe <Boolean>

Shows or hides the info table or queries its visiblity.

Its position can be defined using `DISPlay:ANNotation:INFO:POSition`.

Parameters:
<Boolean> ON (1) | OFF (0)
*RST: OFF

Manual operation: See "Info Table: Show / Position" on page 803

DISPlay:CMAP:BWSCheme[:STATe] <Boolean>
DISPlay:CMAP:LSCHeme[:STATe] <Boolean>
DISPlay:CMAP:LSSCheme[:STATe] <Boolean>

Applies the "Black White Scheme", "Line Styles Scheme", and "Light Scheme" modifiers to the user color scheme or removes them.

Parameters:
<Boolean>
ON (1)
Apply modifier
OFF (0)
Remove modifier

Manual operation: See "General > Black White Scheme / Line Styles Scheme / Light Scheme" on page 806

DISPlay:CMAP:LIMit:FCOLorize[:STATe] <Boolean>

Assigns a different trace color to failed trace segments ("Colorize Trace when Failed").

Parameters:
<Boolean> ON | OFF - colorize trace or keep original trace color.
*RST: n/a (a *RST does not affect the setting). In the factory configuration, OFF is preset.

Example: See `DISPlay:CMAP:LIMit[:STATe]`

Manual operation: See "Limit Test > Colorize Trace when Failed" on page 805

DISPlay:CMAP:LIMit:FSYMbol[:STATe] <Boolean>

Displays or hides the limit fail symbols (colored squares) on the trace.

Parameters:
<Boolean> ON | OFF - show or hide symbols.
*RST: n/a (a *RST does not affect the setting). In the factory configuration, ON is preset.

Example: See `DISPlay:CMAP:LIMit[:STATe]`

Manual operation: See "Limit Test > Show Limit Fail Symbols" on page 804

DISPlay:CMAP:LIMit[:STATe] <Boolean>

Displays all limit lines either with individually configured colors or with the color of the associated trace(s). The colors of all display elements are defined via `DISPlay:CMAP<DispEl>:RGB`.

.

Parameters:

<Boolean> ON - the limit line colors are defined via `DISPlay:CMAP<DispEl>:RGB` where `<DispEl>` = 9 ... 12.
The limit line colors are independent of the trace colors.
OFF - all limit lines have the color of the associated trace.

*RST: n/a (a *RST does not affect the setting). In the factory configuration, OFF is preset.

Example: `DISP:CMAP:LIMit OFF`
Use the trace colors for all limit lines associated with each trace. Subsequent limit line color definitions will be ignored until individual limit settings are enabled again.
`DISPlay:CMAP:LIMit:FCOLorize:STATe ON`
Assign a different trace color to failed trace sections.
`DISPlay:CMAP:LIMit:FSYMbol:STATe OFF`
Remove the limit fail symbols from the trace.

Manual operation: See "Limit Test > Use Trc Color for Limit Lines" on page 805

DISPlay:CMAP:EYE:TCHigh:RGB <arg0>, <arg1>, <arg2>
DISPlay:CMAP:EYE:TCLow:RGB <Red>, <Green>, <Blue>

Defines the color gradient of the eye diagram (as a heat map).

- `THHigh` is the "Trace Cumulative High Color", i.e. the color that is used for the most frequent occurrences.
- `TCLow` is the "Trace Cumulative Low Color", i.e. the color that is used for the occurrence value 1.

No occurrence (value 0) is always displayed fully transparent with the background being visible.

Parameters:

<Red>, <Green>, <Blue> RGB values (decimal integers)

Range: 0, ..., 255

Options: R&S ZNB-K20

Manual operation: See "Define Colors" on page 386

DISPlay:CMAP:MARKer[:STATe] <Boolean>

Displays all markers with the same color or display each marker with the color of the associated trace. The colors of all display elements are defined via `DISPlay:CMAP<DispEl>:RGB <Red>, <Green>, <Blue> ...`

Parameters:

<Boolean> ON - all markers have the same color, to be defined via `DISPlay:CMAP<DispEl>:RGB <Red>, <Green>, <Blue>`.
The marker color is independent of the trace colors.
OFF - each marker has the color of the associated trace.

Example: See `DISPlay:CMAP<DispEl>:RGB`

Manual operation: See "General > Same Color all Markers" on page 805

DISPlay:CMAP<DispEl>:RGB <Red>, <Green>, <Blue>[, <TraceStyle>[, <TraceWidth>]]

Defines the user color scheme using the Red/Green/Blue color model. For traces also the line style and width can be defined.

Use `DISPlay:CMAP:TRACe:RGB` to define the properties of a particular trace, referenced by its name.

Suffix:

<DispEl> Number of the display element as described in the table below.

Parameters:

<Red> Red content of the defined color.

Range: 0 (zero intensity, corresponding to a 0 in the 24-bit color model) to 1 (full intensity, corresponding to 255 in the 24-bit color model).

<Green> Green content of the defined color.

Range: 0 (zero intensity, corresponding to a 0 in the 24-bit color model) to 1 (full intensity, corresponding to 255 in the 24-bit color model).

<Blue> Blue content of the defined color.

Range: 0 (zero intensity, corresponding to a 0 in the 24-bit color model) to 1 (full intensity, corresponding to 255 in the 24-bit color model).

<TraceStyle> SOLid | DASHed | DOTTed | DDOTted | DDDotted

Optional trace style, only for traces (`<DispEl>` > 12): One of the enum constants `SOLid | DASHed | DOTTed | DDOTted | DDDotted`.

<TraceWidth> Optional trace width, only for traces (`<DispEl>` > 12).

Range: 1 to 20

Example:

```
*RST; :DISP:CMAP:MARK ON; :CALC:MARK ON
```
Create diagram area no. 1 (with default trace showing the S-parameter S_{21}) and a marker M1.

```
CALC:PAR:SDEF 'Trc2', 'S11'
DISP:WIND:TRAC2:FEED 'TRC2'
```
Create a trace named Trc2 and display the trace in diagram area no. 1. Note that the new trace automatically becomes the active trace.

```
CALC:MARK2 ON
```
Assign a marker M2 to the trace. Both markers are displayed with the same color.

```
DISP:CMAP13:RGB 1,0,0; :DISP:CMAP14:RGB 0,1,0
```
Color the first trace red, the second trace green.

```
DISP:CMAP6:RGB?
```
Query the marker color. The marker color depends on the settings made in previous sessions; it is not reset. A possible response is `0,0,0` for black markers.

```
DISP:CMAP:MARK OFF
```
Change the marker colors: M1 turns red, M2 turns green.

Manual operation: See "Element" on page 804

The numeric suffixes `<DispEl>` denote the following display elements:

<DispEl>	Display Element
1	Background
2	Text
3	Selected Text
4	Grid
5	Reference Line
6	Same Color for all Markers
7	Vertical Range Lines
8	Diagram Title
9	Limit Fail Trace Color
10	Limit Line Type Off
11	Limit Line Type Upper
12	Limit Line Type Lower
13 to 12+N	Trace properties 1 to N, where N is the number of trace colors configured using `SYSTem:DISPlay:TRACes:CCOunt`

DISPlay:CMAP:TRACe:COLor[:STATe] <Boolean>

Defines the trace color schemes in different diagram areas.

Parameters:

<Boolean> OFF - independent color scheme in new diagram area. Moved traces change their color.
ON - color scheme in new diagram area continues the previous color scheme. Moved traces keep their color.

Example:

```
*RST; :DISP:CMAP13:RGB 1,0,0
```
Create diagram area no. 1 (with default trace showing the S-parameter S_{21}) and color the trace red.

```
DISP:CMAP:TRAC:COL OFF; :DISP:WIND2:STAT ON
```
Select independent color schemes for new diagram areas. Create a new diagram area no. 2.

```
CALC:PAR:SDEF 'Trc2', 'S11'; :DISP:WIND2:TRAC2:FEED 'TRC2'
```
Create a new trace named Trc2 and display the trace in a new diagram area no. 2. The new trace is red like the first trace.

```
DISP:CMAP:TRAC:COL ON; :DISP:WIND3:STAT ON
```
Continue the same color scheme in new diagram areas. Create a new diagram area no. 3.

```
CALC:PAR:SDEF 'Trc3', 'S22'; :DISP:WIND3:TRAC3:FEED 'Trc3'
```
Create a new trace named Trc3 and display the trace in a new diagram area no. 3. The new trace is not red.

Manual operation: See "General > Trace Colors per Diagram" on page 805

DISPlay:CMAP:TRACe:RGB <TraceName>, <Red>, <Green>, <Blue>[, <TraceStyle>[, <TraceWidth>]]

Defines the color, style and width of a trace referenced by its name, based on the Red/Green/Blue color model. Use the generalized command `DISPlay:CMAP<DispEl>:RGB` to define the color of other display elements.

Parameters:

<TraceName> Trace name, string parameter

<Red> Red content of the defined color.

Range: 0 (zero intensity, corresponding to a 0 in the 24-bit color model) to 1 (full intensity, corresponding to 255 in the 24-bit color model).

<Green> Green content of the defined color.

Range: 0 (zero intensity, corresponding to a 0 in the 24-bit color model) to 1 (full intensity, corresponding to 255 in the 24-bit color model).

<Blue> Blue content of the defined color.

Range: 0 (zero intensity, corresponding to a 0 in the 24-bit color model) to 1 (full intensity, corresponding to 255 in the 24-bit color model).

<TraceStyle> SOLid | DASHed | DOTTed | DDOTted | DDDotted
Optional trace style, only for traces (`<DispEl>` > 12): One of the string parameters `SOLid | DASHed | DOTTed | DDOTted | DDDotted`.

<TraceWidth> Optional trace width, only for traces (`<DispEl>` > 12).
Range: 1 to 20

Example: `*RST; :DISP:CMAP:TRAC:RGB 'Trc1', 1, 0, 0`
Color the default trace 'Trc1' red.
See also `DISPlay:CMAP<DispEl>:RGB`

Manual operation: See "Properties" on page 804

DISPlay:IWINdow:BFILter[:STATe] <TraceName>, <BandfilterItem>, <Boolean>

Defines the bandfilter search results to be displayed in the Info Window.

Parameters:
<Boolean> 0 – Result <BandfilterItem> of trace <TraceName> is displayed
1 – Result <BandfilterItem> of trace <TraceName> is hidden

Setting parameters:
<TraceName> Name of the related trace.

<BandfilterItem> CENTer | BANDwidth | UPPer | LOWer | LOSS | QFACtor | QFA3 | HEADer
Bandfilter search result (see Display elements of a diagram)

Manual operation: See "Content Selection" on page 807

DISPlay:IWINdow:MARKer<Mk>[:STATe] <TraceName>, <Boolean>

Defines the markers to be displayed in the Info Window.

Suffix:
<Mk> Marker number.

Parameters:
<Boolean> 0 – Marker <Mk> of trace <TraceName> is displayed
1 – Marker <Mk> of trace <TraceName> is hidden

Setting parameters:
<TraceName> Name of the related trace.

Manual operation: See "Content Selection" on page 807

DISPlay:IWINdow[:STATe] <Boolean>

Shows/hides the Info Window

Parameters:

<Boolean> 0 – hide the "Info Window" (default)
1 – show the "Info Window"

Manual operation: See "Info Window" on page 803

DISPlay:LAYout <LayoutMode>

Arranges the diagrams in the screen, leaving the diagram contents unchanged.

Parameters:

<LayoutMode> LINeup | STACk | HORizontal | VERTical | GRID

LINeup – the diagrams are arranged side by side.
STACk – the diagrams are arranged one on top of the other.
HORizontal – the diagrams are arranged in horizontal rows.
VERTical – the diagrams are arranged in vertical rows.
GRID – the diagrams are arranged as a rectangular matrix. The number of rows and columns is as defined with command `DISPlay:LAYout:GRID`.

Example: See `DISPlay:LAYout:GRID`

Manual operation: See "Split Type" on page 800

DISPlay:LAYout:APPLy <LayoutId>

Selects a previously defined layout for display in the analyzer screen.

Parameters:

<LayoutId> Integer value 1, 2 ...

Current number, as defined by `DISPlay:LAYout:DEFine`.

Example: See Creating diagrams

Manual operation: See "Additional Functionality: SCPI Commands" on page 800

DISPlay:LAYout:DEFine <LayoutId>, <LayoutFormatMode>, <LayoutData>
DISPlay:LAYout:DEFine? <LayoutId>

Creates a horizontal or vertical display layout and provides it with an identifier (<LayoutId>).

Layouts are defined row by row (horizontal layouts) or column by column (vertical layouts).

- A horizontal layout consists of N rows, each of height h_i (i = 1 to N). The heights are defined in units relative to the total height of the screen, i.e. their sum $h_1 + h_2 + ... h_N$ must be equal to 1.00.
  Each row contains a selectable number of diagrams with independent widths w_{ij} (j = 1, 2 ...M(i)). The sum of the widths in each row must also match the screen width, hence $w_{i1} + w_{i2} + ... w_{iM(i)} = 1.00$ for all rows (i = 1 to N).

The <LayoutData> string for horizontal layouts reads 'h_1,w_{11},w_{12} ... $w_{1M(1)}$;h_2,w_{12},w_{22} ... $w_{2M(2)}$; ... ;h_N, w_{N1},w_{N2} ... $w_{NM(N)}$'.
A semicolon separates different rows, a comma separates different diagram widths within a row.

- The definition of a vertical layout is analogous, however, the role of rows and columns is interchanged.

The query returns the layout data in an alternative, executable format. The executable format is also used by `DISPlay:LAYout:EXECute`.

Use `DISPlay:LAYout:JOIN` or `DISPlay:LAYout:EXECute` to create more complicated (nested) layouts.

Note: The maximum number of diagrams in a layout is 256.

Parameters:

<LayoutFormatMode> HORizontal | VERTical
Horizontal or vertical layout; see above.

<LayoutData> String parameter defining the number of diagrams and their position (easy format); see above.

Parameters for setting and query:

<LayoutId> Integer value 1, 2 ...
Current number, used by other `DISPlay:LAYout...` commands to reference the created layout.

Example: See Creating diagrams

Manual operation: See "Additional Functionality: SCPI Commands" on page 800

DISPlay:LAYout:EXECute <LayoutData>

Creates and displays a horizontal or vertical display layout. The query returns the layout data of the currently displayed layout (the last layout selected via `DISPlay:LAYout:APPLy`) in executable format.

The executable format is an extension of the easy format used by `DISPlay:LAYout:DEFine`.

- The <LayoutData> string consists of two parts: <LayoutData> = '(<StartFormat>,<RepeatFormat$_1$>,<Repeat Format$_2$> ...). The <StartFormat> descriptor distinguishes between horizontal and vertical layouts and defines the number of rows or columns. A <RepeatFormat> descriptor follows for each row or colum in the layout. The <RepeatFormat> descriptors can be nested in order to describe joined layouts; refer to Creating diagrams for an easy example.
- For a horizontal layout with N rows, each of height h_i (i = 1 ... N) and filled with M(i) diagrams with independent widths w_{ij} (j = 1, 2 ...M(i)), the data string is composed as follows:
<StartFormat> = N,1,0.00,0.00
<RepeatFormat$_i$> = (1,M(i),1.00,h_i[w_{i1},1.00], [w_{i2},1.00] ... [$w_{iM(i)}$,1.00])

- For a vertical layout with N columns, each of width w_i (i = 1 ... N) and filled with M(i) diagrams with independent heights h_{ij} (j = 1, 2 ...M(i)), the data string is composed as follows:
  <StartFormat> = 1,N,0.00,0.00
  <RepeatFormat$_i$> = (M(i),1,w_i,1.00,[1.00,h_{i1}], [1.00,h_{i2}] ... [1.00,$h_{iM(i)}$])

Note: The maximum number of diagrams in a layout is 256.

Parameters:

<LayoutData> String parameter defining the number of diagrams and their position (executable format); see above.

Example: See Creating diagrams

Manual operation: See "Additional Functionality: SCPI Commands" on page 800

DISPlay:LAYout:GRID <Rows>, <Columns>

Defines the number of rows and columns if `DISPlay:LAYout GRID` is set.

Parameters:

<Rows> Range: 1 to 16
*RST: 1

<Columns> Range: 1 to 16
*RST: 1

Example: `DISPlay:LAYout GRID`
Select te split type where the diagrams are arranged in rows and columns.
`DISPlay:LAYout:GRID 2,2`
Arrange 4 diagrams in two rows and two columns.

Manual operation: See "Diagrams / Rows / Columns" on page 800

DISPlay:LAYout:JOIN <MainLayoutId>, <DiagramNumber>, <SubLayoutId>

Creates a nested layout, inserting a sub-layout into one of the diagrams of a main layout. Main layout and sub-layout must be defined previously, preferably using `DISPlay:LAYout:DEFine`.

Note: The maximum number of joined levels within a layout is 16.

Setting parameters:

<MainLayoutId> Integer value 1, 2 ...
Current number of main layout, as defined by `DISPlay:LAYout:DEFine`.

<DiagramNumber> Integer value 1, 2 ...
Diagram number in the main layout

<SubLayoutId> Integer value 1, 2 ...
Current number of sub-layout, as defined by `DISPlay:LAYout:DEFine`.
*RST: n/a

Example: See Creating diagrams

Usage: Setting only

Manual operation: See "Additional Functionality: SCPI Commands" on page 800

DISPlay:LAYout:OVERlay

Displays all traces of the active recall set in a single diagram, which is maximized to occupy the whole screen.

Usage: Event

Manual operation: See "Overlay All" on page 798

DISPlay:LAYout:SPLit

Creates a separate diagram for each trace in the active recall set and automatically arranges those diagrams in the diagram area. Existing diagrams are deleted.

Usage: Event

Manual operation: See "Split All" on page 798

DISPlay:MENU:KEY:ACTion:CATalog?

Displays the identifiers of the available dialog opener actions as a comma-separated list of strings.

Use `DISPlay:MENU:KEY:EXECute` to open one of the dialogs.

Usage: Query only

DISPlay:MENU:KEY:EXECute <MenuKey>

Opens the dialog identified by <MenuKey>.

Use `DISPlay:MENU:KEY:ACTion:CATalog?` to display the available opener IDs.

Note that no error is generated if a valid <MenuKey> is specified but the dialog cannot be opended for any other reason.

Setting parameters:
<MenuKey> Identifier of a dialog opener

Example:
```
*RST; DISP:MENU:KEY:EXECute ':Cal:Management:CalConnectorTypes'
```
Activates the Cal Connector Types dialog.

Usage: Setting only

Tip: When working with the GPIB Explorer, switch to raw mode ("Options > Raw mode" in the IECWIN32 GUI) before executing this command.

DISPlay:MENU:KEY:SELect <MenuKey>

Activates the softtool tab with identifier <MenuKey>.

Use `DISPlay:MENU:KEY:TOOL:CATalog?` to display the available identifiers.

Note that no error is generated if a valid <MenuKey> is specified but the tab cannot be activated for any other reason.

Setting parameters:

<MenuKey> Identifier of a softtool tab

Example: `*RST; DISP:MENU:KEY:SEL ':Meas:SParams'`
Activates the "S-Params" tab of the "Meas" softtool.

Usage: Setting only

Tip: When working with the GPIB Explorer, switch to raw mode ("Options > Raw mode" in the IECWIN32 GUI) before executing this command.

DISPlay:MENU:KEY:TOOL:CATalog?

Displays the identifiers of the available softtool tabs as a comma-separated list of strings.

Use `DISPlay:MENU:KEY:SELect` to activate one of the tabs.

Usage: Query only

DISPlay:RFSize <RelFontSize>

Defines the size of the fonts in the diagram on a relative scale.

Parameters:

<RelFontSize> Relative font size

Range: 80 % to 170 %
*RST: 100 %
Default unit: percent

Example: `*RST; :DISP:RFS 80`
Use smaller fonts to gain more space for the traces in the diagram.

Manual operation: See "Font Size" on page 803

DISPlay[:WINDow<Wnd>]:ANNotation:MARKer:INFO[:STATe] <Boolean>

If enabled, the marker info field is displayed.

Suffix:
<Wnd>

Parameters:
<Boolean> *RST: ON (1)

Manual operation: See "Show" on page 524

DISPlay[:WINDow<Wnd>]:CATalog?

Returns the numbers and names of all diagrams in the current recall set.

The response is a string containing a comma-separated list of diagram area numbers and names, see example below. If all diagram areas have been deleted, the response is an empty string ("").

Suffix:
<Wnd> Number of a diagram. This suffix is ignored; the command returns a list of all diagrams.

Example:
```
*RST; :DISP:WIND2:STAT ON
```
Create diagram no. 2.
```
DISP:WIND2:NAME 'S11 Test Diagram'
```
Assign a name to the new diagram.
```
DISP:CAT?
```
Query all diagrams and their names. As a default diagram no. 1 is created upon *RST, the response is `''1,1,2,S11 Test Diagram'`. The first diagram is not named; its default name is equal to the diagram number.
```
CALC:PAR:SDEF 'Win2_Tr1', 'S11'
```
Create a trace named Win2_Tr1 to measure the input reflection coefficient S_{11}.
```
DISP:WIND2:TRAC9:FEED 'Win2_Tr1'
```
Display the generated trace in diagram area no. 2, assigning the trace number 9 to it.
```
DISP:WIND2:TRAC:CAT?
```
Query all traces in diagram area no. 2. The response is `'9,Win2_Tr1'`.

Usage: Query only

Manual operation: See "Title" on page 797

DISPlay[:WINDow<Wnd>]:MARKer:CATalog?

Returns the configured markers and their states (active, inactive).

Suffix:
<Wnd> Number of a diagram. This suffix is ignored; all diagrams are considered.

Usage: Query only

Manual operation: See "Mkr 1 ... Mkr 10" on page 505

DISPlay[:WINDow<Wnd>]:MAXimize <Boolean>

Maximizes all diagram areas in the active recall set or restores the previous display configuration.

Suffix:
<Wnd> Number of the diagram area to become the active diagram area. `DISPlay:WINDow<Wnd>:MAXimize` acts on all diagrams of the current recall set, however, the diagram no. `<Wnd>` is displayed on top of the others.

Parameters:
<Boolean> ON | OFF - maximize all diagram areas or restore the previous display configuration.

*RST: OFF

Example: `*RST; :DISP:WIND2:STAT ON`
Create diagram areas no. 1 (with default trace) and 2 (with no trace).
`DISP:WIND2:MAXimize ON`
Maximize the diagram areas, placing area no. 2 on top.

Manual operation: See "Maximize Diagram" on page 797

DISPlay[:WINDow<Wnd>]:NAME <Name>

Defines a name for diagram area `<Wnd>`. The name appears in the list of diagram areas, to be queried by `DISPlay[:WINDow<Wnd>]:CATalog?`.

Suffix:
<Wnd> Number of the diagram area.

Parameters:
<Name> String variable for the name.

Example: See `DISPlay[:WINDow<Wnd>]:CATalog?`

Manual operation: See "Title" on page 797

DISPlay[:WINDow<Wnd>]:OVERview[:STATe] <Boolean>

Enables the zoom function with an additional overview window for the diagram no. `<Wnd>` or removes the overview window from a diagram.

Suffix:
<Wnd> Number of the zoomed diagram area

Parameters:
<Boolean> ON – activate the zoom window with overview window
OFF – remove the overview window

*RST: OFF

Example: See DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:ZOOM[:STATe]

DISPlay[:WINDow<Wnd>]:SIZE <WindowSize>

Suffix:
<Wnd> Number of a diagram.

Parameters:
<WindowSize> NORMal | MAXimized | MINimized

NORMal
The window has normal size.

MAXimized
The window is maximized.

MAXimized
The window is minimized.

DISPlay[:WINDow<Wnd>][:STATe] <Boolean>

Creates or deletes a diagram area, identified by its area number <Wnd>.

Suffix:
<Wnd> Number of the diagram area to be created or deleted.

Parameters:
<Boolean> ON | OFF - creates or deletes diagram area no. <Wnd>.

*RST: -

Example:
```
CALC4:PAR:SDEF 'Ch4Tr1', 'S11'
```
Create channel 4 and a trace named Ch4Tr1 to measure the input reflection coefficient S_{11}.
```
DISP:WIND2:STAT ON
```
Create diagram area no. 2.
```
DISP:WIND2:TRAC9:FEED 'CH4TR1'
```
Display the generated trace in diagram area no. 2, assigning the trace number 9 to it.

Manual operation: See "Add Tr+Diag" on page 796

DISPlay[:WINDow<Wnd>]:TITLe:DATA <Title>

Defines a title for diagram area <Wnd>.

Suffix:
<Wnd> Number of the diagram area.

Parameters:
<Title> String variable for the title. The length of the title is practically unlimited but should be kept short enough to be displayed in the diagrams.

Example: `*RST; :DISP:WIND:TITL:DATA 'S21 Test Diagram'`
Define a title for the default diagram area. The title is displayed below the top of the diagram area.
`DISP:WIND:TITL OFF; TITL:DATA?`
Hide the title. The title is no longer displayed but still defined so it can be displayed again.

Manual operation: See "Title" on page 797

DISPlay[:WINDow<Wnd>]:TITLe[:STATe] <Boolean>

Displays or hides the title for area number `<Wnd>`, defined by means of `DISPlay:WINDow<Wnd>:TITLe:DATA`.

Suffix:
<Wnd> Number of the diagram area.

Parameters:
<Boolean> ON | OFF - displays or hides the title.
*RST: ON

Example: See `DISPlay[:WINDow<Wnd>]:TITLe:DATA`

Manual operation: See "Show Title" on page 797

DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:CATalog?

Returns the numbers and names of all traces in diagram area no. `<Wnd>`.

Suffix:
<Wnd> Number of a diagram area.

<WndTr> Trace number used to distinguish the traces of the same diagram area `<Wnd>`. This suffix is ignored; the command returns a list of all traces.

Example: See `DISPlay[:WINDow<Wnd>]:CATalog?`

Usage: Query only

Manual operation: See "Active Diagram" on page 796

DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:COUPle <Boolean>[, <TraceName>]

Defines or queries a trace's scale coupling.

Suffix:
<Wnd> Diagram number

<WndTr> Number of a trace within diagram <Wnd>

Parameters:
<Boolean> **OFF (0)**
Uncoupled

ON (1)
Coupled

<TraceName> Trace name; ignored when coupling is turned off.

Example: `:DISPlay:WINDow1:TRACe2:COUPle ON, 'Trc3'`
couples the scale of diagram 1's trace 2 to the scale of the trace named 'Trc3'.
`:DISPlay:WINDow1:TRACe2:COUPle?`
returns 2; 'Trc3'
`:DISPlay:WINDow1:TRACe2:COUPle OFF`
decouples the scale of diagram 1's trace 2

Manual operation: See "Couple All Traces / Couple Trc ... To Trace" on page 447

DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:COUPle:ALL <Boolean>

Couples the scales of diagram area <Wnd>'s traces to the scale of its trace no. <WndTr>, or decouples them.

Suffix:
<Wnd> Diagram number

<WndTr> Number of a trace within diagram <Wnd>

Parameters:
<Boolean>
OFF (0)
Uncoupled
ON (1)
Coupled

Usage: Setting only

Manual operation: See "Couple All Traces / Couple Trc ... To Trace" on page 447

DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:DELete

Releases the assignment between a trace and a diagram area, as defined by means of `DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:FEED <TraceName>` and expressed by the `<WndTr>` suffix. The trace itself is not deleted; this must be done via `CALCulate<Ch>:PARameter:DELete <TraceName>`.

Suffix:
<Wnd> Number of an existing diagram area (defined by means of `DISPlay[:WINDow<Wnd>][:STATe] ON`).

<WndTr> Trace number used to distinguish the traces of the same diagram area `<Wnd>`.

Example: CALC4:PAR:SDEF 'Ch4Tr1', 'S11'
Create channel 4 and a trace named Ch4Tr1 to measure the input reflection coefficient S_{11}.
DISP:WIND2:STAT ON
Create diagram area no. 2.
DISP:WIND2:TRAC9:FEED 'CH4TR1'
Display the generated trace in diagram area no. 2, assigning the trace number 9 to it.
DISP:WIND2:TRAC9:DELete
Release the assignment between trace no. 9 and window no. 2. The trace can still be referenced with its trace name Ch4Tr1.

Usage: Event

DISPlay[:WINDow<Wnd>]:TRACe:EFEed <TraceName>

Assigns an existing trace (CALCulate<Ch>:PARameter:SDEFine <TraceName>) to a diagram area <Wnd>, and displays the trace. Use DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:FEED to assign the trace to a diagram area using a numeric suffix (e.g. in order to use the DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:Y:OFFSet command).

At the GUI, use the Trace Manager dialog to obtain an overview of all channels and traces, including traces that are not displayed.

Note:

Up to FW V3.45, command execution failed for traces that were already assigned to a diagram area, and for memory traces. Starting with FW V3.50 the command simply moves those traces (along with their parent trace, in case of memory traces).

Suffix:
<Wnd> Number of an existing diagram area (defined by means of DISPlay[:WINDow<Wnd>][:STATe] ON).

Setting parameters:
<TraceName> String parameter for the trace name, e.g. 'Trc4'.

Example: CALC4:PAR:SDEF 'Ch4Tr1', 'S11'
Create channel 4 and a trace named Ch4Tr1 to measure the input reflection coefficient S_{11}.
DISP:WIND2:STAT ON
Create diagram area no. 2.
DISP:WIND2:TRAC:EFE 'CH4TR1'
Display the generated trace in diagram area no. 2. No trace number is assigned.

Usage: Setting only

DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:FEED <TraceName>

Assigns an existing trace (`CALCulate<Ch>:PARameter:SDEFine`) to a diagram area, using the `<WndTr>` suffix, and displays the trace. Use `DISPlay[:WINDow<Wnd>]:TRACe:EFEed` to assign the trace to a diagram area without using a numeric suffix.

Tip: A trace can be assigned to a diagram only once. If a attempt is made to assign the same trace a second time (e.g. by typing `DISP:WIND2:TRAC8:FEED 'CH4TR1'` after executing the program example below) an error message -114,"Header suffix out of range" is generated. You can open the "Trace Manager" dialog to obtain an overview of all channels and traces, including the traces that are not displayed.

Suffix:

<Wnd> Number of an existing diagram area (defined by means of `DISPlay[:WINDow<Wnd>][:STATe] ON`).

<WndTr> Trace number used to distinguish the traces of the same diagram area `<Wnd>`.

Parameters:

<TraceName> String parameter for the trace name, e.g. 'Trc4'.

Example:

```
CALC4:PAR:SDEF 'Ch4Tr1', 'S11'
```

Create channel 4 and a trace named Ch4Tr1 to measure the input reflection coefficient S_{11}.

```
DISP:WIND2:STAT ON
```

Create diagram area no. 2.

```
DISP:WIND2:TRAC9:FEED 'CH4TR1'
```

Display the generated trace in diagram area no. 2, assigning the trace number 9 to it.

Manual operation: See "Diagram" on page 451

DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:LABel:SHOW <TraceName>[, <Boolean>]

Displays or hides the label of an existing trace, identified by its `<TraceName>`.

Suffix:

<Wnd> ignored

<WndTr> ignored

Parameters:

<TraceName> Trace name (string parameter), e.g. 'Trc4'

<Boolean> ON | OFF – display or hide the label of the related trace

Example:

```
*RST; :DISP:TRAC:LAB:SHOW? 'Trc1'
```

Reset the analyzer, creating the default trace 'Trc1'. The trace label is displayed; the query returns 1.

```
DISP:TRAC:LAB:SHOW 'Trc1', OFF
```

Hide the label of trace 'Trc1'.

Manual operation: See "Table Area" on page 453

DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:SHOW <TraceName>[, <Boolean>]

Displays or hides an existing trace, identified by its trace name `<Trace_Name>`, or a group of traces.

Tip: You can open the trace manager to obtain an overview of all channels and traces, including the traces that are not displayed.

Suffix:

<Wnd> Number of a diagram area. This suffix is ignored; the command affects traces in all diagram areas.

<WndTr> Trace number. This suffix is ignored; the trace is referenced by its name.

Parameters:

<TraceName> DALL – all data traces
MALL – all memory traces
<string> – single trace identified by its trace name (string parameter), e.g. 'Trc4'.

<Boolean> ON | OFF – display or hide traces.

Example:

```
*RST; :DISP:TRAC:SHOW? 'Trc1'
```
Reset the analyzer, creating the default trace 'Trc1'. The trace is displayed; the query returns 1.

```
CALC4:PAR:SDEF 'Ch4Tr1', 'S11'
```
Create channel 4 and a trace named Ch4Tr1 to measure the input reflection coefficient S_{11}.

```
DISP:WIND2:STAT ON; :DISP:WIND2:TRAC:FEED
'CH4TR1'
```
Create diagram area no. 2 and display the generated trace in the diagram area.

```
DISP:TRAC:SHOW DALL, OFF
```
Hide both traces in both diagrams.

```
DISP:TRAC:SHOW? DALL
```
Query whether all data traces are displayed. The response 0 means that at least one trace is hidden.

Manual operation: See "Show" on page 451

DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:X:OFFSet <StimulusOffset>

Shifts the trace `<WndTr>` in horizontal direction, leaving the positions of all markers unchanged.

Suffix:

<Wnd> Number of an existing diagram area (defined by means of `DISPlay[:WINDow<Wnd>][:STATe] ON`).

<WndTr> Existing trace number, assigned by means of `DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:FEED`.

Parameters:

<StimulusOffset> Stimulus offset value.
The range and unit depends on the sweep type.
Default unit: NN

Example: `*RST; :DISP:WIND:TRAC:X:OFFS 1MHZ; :DISP:WIND:TRAC:Y:OFFS 10`
Create the default trace and shift it horizontally by 1 MHz, vertically by 10 dB.

Manual operation: See "Stimulus" on page 473

DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:Y:OFFSet <MagnitudeFactor>[, <PhaseFactor>[, <RealPart>[, <ImaginaryPart>]]]

Modifies all points of the trace `<WndTr>` by means of an added and/or a multiplied complex constant. The response values M of the trace are transformed according to:

Suffix:

<Wnd> Number of an existing diagram area (defined by means of `DISPlay[:WINDow<Wnd>][:STATe] ON`).

<WndTr> Existing trace number, assigned by means of `DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:FEED`.

Parameters:

<MagnitudeFactor> Multiplied magnitude factor

Range: -300 dB to + 300 dB
*RST: 0 dB
Default unit: dB

<PhaseFactor> Multiplied phase factor, optional for setting command but returned by query

Range: -3.4*1038 deg to +3.4*1038 deg
*RST: 0 deg
Default unit: deg

<RealPart> Real and imaginary part of added complex constant, optional for setting command but returned by query

Range: -3.4*1038 to +3.4*1038
*RST: 0

<ImaginaryPart>

Example: `*RST; :DISP:WIND:TRAC:X:OFFS 1MHZ; :DISP:WIND:TRAC:Y:OFFS 10`
Create the default trace and shift it horizontally by 1 MHz, vertically by 10 dB.
`DISP:WIND:TRAC:Y:OFFS?`
Query all response offset values. The response is `10,0,0,0`.

Manual operation: See "Mag / Phase / Real / Imag" on page 473

DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:Y[:SCALe]:AUTO <Activate>[, <TraceName>]

Displays the entire trace in the diagram area, leaving an appropriate display margin. The trace can be referenced either by its number `<WndTr>` or by its name `<TraceName>`.

Suffix:

<Wnd> Number of an existing diagram area (defined by means of `DISPlay[:WINDow<Wnd>][:STATe] ON`). This suffix is ignored if the optional `<TraceName>` parameter is used.

<WndTr> Existing trace number, assigned by means of `DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:FEED`. This suffix is ignored if the optional `<TraceName>` parameter is used.

Setting parameters:

<Activate> ONCE

Activate the autoscale function.

<TraceName> Optional string parameter for the trace name, e.g. 'Trc4'. If this optional parameter is present, both numeric suffixes are ignored (trace names must be unique across different channels and windows).

Example:

```
*RST; DISP:WIND:TRAC:Y:PDIV?; RLEV?
```

Query the value between two grid lines and the reference value for the default trace. The response is `10;0`.

```
DISP:WIND:TRAC:Y:AUTO ONCE; PDIV?; RLEV?
```

or:

```
DISP:WIND:TRAC:Y:AUTO ONCE, 'Trc1'; PDIV?;
RLEV?
```

Autoscale the default trace and query the scaling parameters again. In general both values have changed.

Usage: Setting only

Manual operation: See "Auto Scale Trace" on page 445

DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:Y[:SCALe]:BOTTom <LowEdge>[, <TraceName>]
DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:Y[:SCALe]:TOP <UppEdge>[, <TraceName>]

These commands define the lower (bottom) and upper (top) edge of the diagram area `<Wnd>`.

Suffix:

<Wnd> Number of an existing diagram area (defined by means of `DISPlay[:WINDow<Wnd>][:STATe] ON`). This suffix is ignored if the optional `<TraceName>` parameter is used.

<WndTr> Existing trace number, assigned by means of `DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:FEED`. This suffix is ignored if the optional `<TraceName>` parameter is used.

Parameters:

<UppEdge> Value and unit for the lower or upper diagram edge. Range and unit depend on the measured quantity, see "Units for DISPlay... commands" on page 1211.

Default unit: NN

<TraceName> Optional string parameter for the trace name, e.g. 'Trc4'. If this optional parameter is present, both numeric suffixes are ignored (trace names must be unique across different channels and windows).

Example:

```
CALC4:PAR:SDEF 'Ch4Tr1', 'S11'
```

Create channel 4 and a trace named Ch4Tr1 to measure the input reflection coefficient S_{11}.

```
DISP:WIND2:STAT ON
```

Create diagram area no. 2.

```
DISP:WIND2:TRAC9:FEED 'CH4TR1'
```

Display the generated trace in diagram area no. 2, assigning the trace number 9 to it.

```
DISP:WIND2:TRAC9:Y:BOTT -40; TOP 10
```

or:

```
DISP:WIND2:TRAC:Y:BOTT -40, 'CH4TR1'; TOP 10, 'CH4TR1'
```

Scale the diagram between -40 dB and +10 dB.

Manual operation: See "Max / Min" on page 446

DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:Y[:SCALe]:CONTinuous <Boolean>

Enables or disables continuous auto-scaling for trace <WndTr> of diagram area <Wnd>.

Suffix:

<Wnd> Diagram number

<WndTr> Number of a trace within diagram <Wnd>

Parameters:

<Boolean> **OFF (0)**
Continuous auto-scaling disabled (default)

ON (1)
Continuous auto-scaling enabled

Manual operation: See "Auto Scale Tr. Cont." on page 446

DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:Y[:SCALe]:PDIVision <VerticalDiv>[, <TraceName>]

Sets the value between two grid lines (value “per division”) for the diagram area <Wnd>. When a new PDIVision value is entered, the current RLEVel is kept the same, while the top and bottom scaling is adjusted for the new PDIVision value.

Suffix:

<Wnd> Number of an existing diagram area (defined by means of DISPlay[:WINDow<Wnd>][:STATe] ON). This suffix is ignored if the optional <TraceName> parameter is used.

<WndTr> Existing trace number, assigned by means of DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:FEED. This suffix is ignored if the optional <TraceName> parameter is used.

Parameters:

<VerticalDiv> Value and unit for the vertical diagram divisions. Range and unit depend on the measured quantity, see "Units for DISPlay... commands" on page 1211.

Default unit: NN

<TraceName> Optional string parameter for the trace name, e.g. 'Trc4'. If this optional parameter is present, both numeric suffixes are ignored (trace names must be unique across different channels and windows).

Example:

```
CALC4:PAR:SDEF 'Ch4Tr1', 'S11'
```

Create channel 4 and a trace named Ch4Tr1 to measure the input reflection coefficient S_{11}.

```
DISP:WIND2:STAT ON
```

Create diagram area no. 2.

```
DISP:WIND2:TRAC9:FEED 'CH4TR1'
```

Display the generated trace in diagram area no. 2, assigning the trace number 9 to it.

```
DISP:WIND2:TRAC9:Y:PDIV 5
```

or:

```
DISP:WIND2:TRAC:Y:PDIV 5, 'CH4TR1'
```

Set the value per division to 5 dB.

Manual operation: See "Scale/Div" on page 445

DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:Y[:SCALe]:RLEVel <RefLevel>[, <TraceName>]

Sets the reference level (or reference value) for a particular displayed trace. Setting a new reference level does not affect the value of PDIVision. The trace can be referenced either by its number <WndTr> or by its name <TraceName>.

Suffix:

<Wnd> Number of an existing diagram area (defined by means of `DISPlay[:WINDow<Wnd>][:STATe] ON`). This suffix is ignored if the optional `<TraceName>` parameter is used.

<WndTr> Existing trace number, assigned by means of `DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:FEED`. This suffix is ignored if the optional `<TraceName>` parameter is used.

Parameters:

<RefLevel> Value and unit for the reference level (or reference value, if the trace does not show a level). Range and unit depend on the measured quantity, see "Units for DISPlay... commands" on page 1211.

Default unit: NN

<TraceName> Optional string parameter for the trace name, e.g. 'Trc4'. If this optional parameter is present, both numeric suffixes are ignored (trace names must be unique across different channels and windows).

Example:

```
CALC4:PAR:SDEF 'Ch4Tr1', 'S11'
```

Create channel 4 and a trace named Ch4Tr1 to measure the input reflection coefficient S_{11}.

```
DISP:WIND2:STAT ON
```

Create diagram area no. 2.

```
DISP:WIND2:TRAC9:FEED 'CH4TR1'
```

Display the generated trace in diagram area no. 2, assigning the trace number 9 to it.

```
DISP:WIND2:TRAC9:Y:RLEV -10
```

or:

```
DISP:WIND2:TRAC:Y:RLEV -10, 'CH4TR1'
```

Change the reference level to -10 dB.

Manual operation: See "Ref Value" on page 446

DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:Y[:SCALe]:RPOSition <RefPosition>[, <TraceName>]

Sets the point on the y-axis to be used as the reference position as a percentage of the length of the y-axis. The reference position is the point on the y-axis which should equal the `RLEVel`.

Suffix:

<Wnd> Number of an existing diagram area (defined by means of `DISPlay[:WINDow<Wnd>][:STATe] ON`). This suffix is ignored if the optional `<TraceName>` parameter is used.

<WndTr> Existing trace number, assigned by means of `DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:FEED`. This suffix is ignored if the optional `<TraceName>` parameter is used.

Parameters:

<RefPosition> Value of the reference position in percent. The top of the y-axis is defined to have a reference position of 100%, while the bottom of the y-axis is defined to have a reference position of 0%.

Range: 0% to 100%
*RST: 80%
Default unit: %

<TraceName> Optional string parameter for the trace name, e.g. 'Trc4'. If this optional parameter is present, both numeric suffixes are ignored (trace names must be unique across different channels and windows).

Example:

```
CALC4:PAR:SDEF 'Ch4Tr1', 'S11'
```

Create channel 4 and a trace named Ch4Tr1 to measure the input reflection coefficient S_{11}.

```
DISP:WIND2:STAT ON
```

Create diagram area no. 2.

```
DISP:WIND2:TRAC9:FEED 'CH4TR1'
```

Display the generated trace in diagram area no. 2, assigning the trace number 9 to it.

```
DISP:WIND2:TRAC9:Y:RPOS 50
```

or:

```
DISP:WIND2:TRAC:Y:RPOS 50, 'CH4TR1'
```

Set the reference position to the center of the diagram area.

Manual operation: See "Ref Pos" on page 446

DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:ZOOM:BOTTom <LowEdge>[, <TraceName>]
DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:ZOOM:TOP <UppEdge>[, <TraceName>]

These commands specify the lower and upper edge of the zoom window. In contrast to manual control, all or part of the zoom window may be outside the original diagram.

Suffix:

<Wnd> Number of an existing diagram (defined by means of `DISPlay[:WINDow<Wnd>][:STATe] ON`). This suffix is ignored if the optional `<TraceName>` parameter is used.

<WndTr> Existing trace number, assigned by means of `DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:FEED`. This suffix is ignored if the optional `<TraceName>` parameter is used.

Parameters:

<UppEdge> Lower or upper edge of the zoom window. Range and unit depend on the measured quantity, see "Units for DISPlay... commands" on page 1211.

Default unit: NN

<TraceName> Optional string parameter for the trace name, e.g. 'Trc4'. If this optional parameter is present, both numeric suffixes are ignored (trace names must be unique across different channels and windows).

Example: See `DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:ZOOM[:STATe]`

Manual operation: See "Max / Min / Start / Stop" on page 449

DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:ZOOM:STARt <LeftBorder>[, <TraceName>]
DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:ZOOM:STOP <RightBorder>[, <TraceName>]

These commands specify the start and stop values of the zoom window (left and right border), respectively. In contrast to manual control, all or part of the zoom window may be outside the original diagram. The range of possible values depends on the R&S ZNB's frequency range; see Section 7.3.14.9, "[SENSe:]FREQuency...", on page 1397.

Suffix:

<Wnd> Number of an existing diagram (defined by means of `DISPlay[:WINDow<Wnd>][:STATe] ON`). This suffix is ignored if the optional `<TraceName>` parameter is used.

<WndTr> Existing trace number, assigned by means of `DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:FEED`. This suffix is ignored if the optional `<TraceName>` parameter is used.

Parameters:

<RightBorder> Left or right border of the zoom window.

Range: See description above.
*RST: Start or stop of the analyzer's sweep range.
Default unit: NN

<TraceName> Optional string parameter for the trace name, e.g. 'Trc4'. If this optional parameter is present, both numeric suffixes are ignored (trace names must be unique across different channels and windows).

Example: See `DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:ZOOM[:STATe]`

Manual operation: See "Max / Min / Start / Stop" on page 449

DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:ZOOM[:STATe] <Boolean>[, <TraceName>]

Applies or disables the zoom function based on the current zoom window settings.

Suffix:

<Wnd> Number of an existing diagram (defined by means of `DISPlay[:WINDow<Wnd>][:STATe] ON`). This suffix is ignored if the optional `<TraceName>` parameter is used.

<WndTr> Existing trace number, assigned by means of `DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:FEED`. This suffix is ignored if the optional `<TraceName>` parameter is used.

Parameters:

<Boolean> Enable or disable the zoom. OFF also restores the original diagram size after a zoom function was applied.

OFF

<TraceName> Optional string parameter for the trace name, e.g. 'Trc4'. If this optional parameter is present, both numeric suffixes are ignored (trace names must be unique across different channels and windows).

Example:

```
*RST; :DISPlay:WINDow1:TRACe:ZOOM:BOTTom -20;
TOP +10; STARt 1GHz; STOP 1.5 GHz
```

Define a zoom rectangle in the default diagram no. 1 ranging from -20 dB to +10 dB and from 1 GHz to 1.5 GHz.

```
DISPlay:WINDow1:TRACe:ZOOM ON
```

Zoom into the diagram no. 1 so that the zoom window fills the entire diagram. The actual sweep range and the stimulus values of the sweep points are not affected.

```
DISPlay:WINDow1:OVERview:STATe ON
```

Activate an additional overview window in the upper part of the diagram.

Manual operation: See "Zoom Select/Stim. Zoom Select" on page 448

DISPlay[:WINDow<Wnd>]:Y[:SCALe]:AUTO [<Common>]

Auto-scales all traces in a diagram area.

Suffix:

<Wnd> Number of an existing diagram area (defined by means of `DISPlay[:WINDow<Wnd>][:STATe] ON`). This suffix is ignored if the optional `<TraceName>` parameter is used.

Setting parameters:

<Common> COMMon

If specified, equally formatted traces are scaled together.

Usage: Setting only

Manual operation: See "Auto Scale Diagram" on page 445

7.3.6 FORMat commands

The FORMAT... commands select a data format for transferring numeric data (including arrays) from and to the analyzer.

FORMat:BORDer 1241
FORMat[:DATA] 1241

FORMat:BORDer <ByteOrder>

Defines how binary data are transferred and interpreted (byte order or endianness).

The firmware adjusts the byte order with the selected remote language (SYSTem:LANGuage).

Parameters:

<ByteOrder> NORMal | SWAPped

NORMal
Most significant byte first (big endian).
Default for remote languages 'ENA' | 'E5071' | 'PNA' | 'HP8510' | 'HP8530' | 'HP8714' | 'HP8720' | 'HP8753'

SWAPped
Least significant byte first (little endian).
Default for remote languages 'SCPI' | 'ZVABT' | 'ZVR'

Example: FORM:BORD NORM
Change the byte order to normal mode.

Manual operation: See "Define *IDN + *OPT..." on page 843

FORMat[:DATA] <TransferFormat>[, <Length>]

Selects the format for numeric data transferred to and from the analyzer.

Note: The format setting is only valid for commands and queries whose description states that the response is formatted as described by FORMat[:DATA]. In particular, it affects trace data transferred by means of the commands in the TRACe:... system.

Parameters:

<TransferFormat> ASCii | REAL

ASCii - numeric data is transferred as ASCII bytes. The numbers are separated by commas as specified in IEEE 488.2.
REAL - Data is transferred in a definite length block as IEEE floating point numbers of the specified <Length>. See Section 6.2.3.5, "Block data format", on page 881.
Note: If binary data is transferred to the analyzer, the receive terminator should be set to EOI (SYSTem:COMMunicate:GPIB[:SELF]:RTERminator EOI) to avoid inadvertent interruption of the data transfer.

<Length> The optional <Length> parameter is needed for REAL format only. It defines the length of the floating point numbers in bits. Valid values are 32 and 64.

*RST: ASCII. The default length of REAL data is 32 bits (single precision).

Example:
```
FORM REAL, 32
```
Select real data format.
```
SYST:COMM:GPIB:RTER EOI
```
Set the terminator to EOI.
```
(During a calibration) ... CORR:CDAT?
'REFLTRACK',1,0
```
Query a system error correction term. The data is transferred in a definite length block which can be written to a file; the analyzer displays the message "<no> bytes binary data received".

7.3.7 HCOPy commands

The `HCOPy...` commands control the output of screen information to an external device. Part of the functionality of this system is included in the "File" menu.

HCOPy:DESTination 1242
HCOPy:DEVice:LANGuage 1243
HCOPy[:IMMediate] 1243
HCOPy:ITEM:ALL 1244
HCOPy:ITEM:LOGO[:STATe] 1244
HCOPy:ITEM:MLISt[:STATe] 1244
HCOPy:ITEM:TIME[:STATe] 1245
HCOPy:PAGE:COLor 1245
HCOPy:PAGE:MARGin:BOTTom 1246
HCOPy:PAGE:MARGin:LEFT 1246
HCOPy:PAGE:MARGin:RIGHt 1246
HCOPy:PAGE:MARGin:TOP 1246
HCOPy:PAGE:ORIentation 1247
HCOPy:PAGE:WINDow 1247

HCOPy:DESTination <PrinterName>

Selects a printer name or file as destination for the screen output.

Parameters:

<PrinterName> String variable containing the printer name. One of the printers accessible from your PC. The following strings are supported in addition:
'MMEM' - print to file. The file name is defined via `MMEMory:NAME`. The command `HCOPy:DEVice:LANGuage` selects the file format.
'DEFPRT' - use default printer, to be selected in the "Devices and Printers" dialog of the Windows control panel.

*RST: n/a (*RST does not overwrite the printer destination)

Example:
```
MMEM:NAME 'C:\Screenshots\PLOT1.BMP'
```
Define a printer file name (without creating the file), assuming that `.BMP` is the current file format (see `HCOPy:DEVice:LANGuage`).
```
HCOP:DEST 'MMEM'; :HCOP
```
Select 'Print to file' and create the printer file specified before.

Manual operation: See "To File..." on page 784

HCOPy:DEVice:LANGuage <Format>

Selects a file format for printer files. Selecting the format is recommended to ensure that the file defined via `MMEMory:NAME` can be read or imported by an external application.

Parameters:

<Format> BMP | PNG | JPG | PDF | SVG

BMP - Windows bitmap
JPG - JPEG bitmap
PNG - portable network graphics format
PDF - portable document format (Adobe® Systems)
SVG - scalable vector graphics format, XML-based

*RST: n/a (*RST does not affect the printer configuration)

Example:
```
HCOP:DEV:LANG BMP
```
Select Windows bitmap format for printer files.
```
MMEM:NAME 'C:\Screenshots\PLOT1.BMP'
```
Define a printer file name and specify an existing directory (without creating the file).
```
HCOP:DEST 'MMEM'; :HCOP
```
Select 'Print to file' and create the printer file specified before.

HCOPy[:IMMediate]

Initializes the print according to the current `HCOPy...` configuration.

Example: `HCOP:DEST '<Printer_name>'`
Select the printer for the output of screen data.
`HCOP`
Start printing.

Usage: Event

Manual operation: See "Print" on page 784

HCOPy:ITEM:ALL

Selects the complete screen contents to be printed, including the logo (`HCOPy:ITEM:LOGO[:STATe]`), time (`HCOPy:ITEM:TIME[:STATe]`), and the marker list (`HCOPy:ITEM:MLISt[:STATe]`).

Example: `HCOP:ITEM:ALL`
Select the complete information to be printed.
`HCOP`
Start printing.

Usage: Event

HCOPy:ITEM:LOGO[:STATe] <Boolean>

Qualifies whether or not the printed output contains the logo. The default R&S logo (file `Logo.gif`) is stored in the `Resources\Images` subdirectory of the VNA program directory and can be replaced by another logo.

Parameters:
<Boolean> ON | OFF - logo is included or excluded.
*RST: n/a (*RST does not affect the printer configuration)

Example: `HCOP:ITEM:ALL`
Select the complete information to be printed.
`HCOP:ITEM:LOGO ON; :HCOP`
Include the logo in the printed output and start printing.

HCOPy:ITEM:MLISt[:STATe] <Boolean>

Qualifies whether or not the printed output contains the information in the marker info field (marker list).

Parameters:
<Boolean> ON | OFF - marker list is included or excluded.
*RST: n/a (*RST does not affect the printer configuration)

Example: `HCOP:ITEM:ALL`
Select the complete information to be printed.
`HCOP:ITEM:MLISt ON; :HCOP`
Include the marker list in the printed output and start printing.

HCOPy:ITEM:TIME[:STATe] <Boolean>

Qualifies whether or not the printed output contains the current date and time.

Parameters:

<Boolean> ON | OFF - date and time is included or excluded.

*RST: n/a (*RST does not affect the printer configuration)

Example: `HCOP:ITEM:ALL`
Select the complete information to be printed.
`HCOP:ITEM:TIME ON; :HCOP`
Include the data and time in the printed output and start printing .

HCOPy:PAGE:COLor <ColorScheme>

Select the color scheme to be used for printing.

Use `SYSTem:DISPlay:COLor` to select the color scheme for on-screen display.

Parameters:

<ColorScheme> UDEFined | DBACkground | LBACkground | BWLStyles | BWSolid | PBWLstyles | PCLBackgrnd | ON | OFF

UDEFined
User-defined print color scheme
Note that there are currently no remote commands to configure the user-defined print color scheme. Use `MMEMory:LOAD:CMAP:HCOPy` to load and activate the user-defined print color scheme from file.

DBACkground
Dark background

LBACkground
Light background

BWLStyles
Black and white with white background and different line styles

BWSolid
Black and white with white background and black solid lines

PBWLstyles
Printer optimized black and white with different line styles

PCLBackgrnd
Printer-optimized color scheme with light background

ON | OFF
For compatibility with previous versions of this command: ON = PCLBackgrnd, OFF = PBWLstyles

*RST: n/a (*RST does not affect the printer configuration)

Example: `HCOP:PAGE:COL PCLBackgrnd`
Select the printer-optimized color scheme with light background.

Manual operation: See "Print Options" on page 786

HCOPy:PAGE:MARGin:BOTTom <LowMargin>

Defines the distance between the bottom of the page and the bottom of the printed information.

Parameters:

<LowMargin> Lower margin

Range: 0.01 mm to 10000 mm
*RST: n/a (*RST does not affect the printer configuration)

Example: `HCOP:PAGE:MARG:BOTT 10; TOP 10`
Set an upper and a lower margin of 1 cm.

HCOPy:PAGE:MARGin:LEFT <LeftMargin>

Defines the distance between the left edge of the page and the left edge of the printed information.

Parameters:

<LeftMargin> Left margin

Range: 0.01 mm to 10000 mm
*RST: n/a (*RST does not affect the printer configuration)

Example: `HCOP:PAGE:MARG:LEFT 10; RIGHT 10`
Set an left and a right margin of 1 cm.

HCOPy:PAGE:MARGin:RIGHt <RightMargin>

Defines the distance between the right edge of the page and the right edge of the printed information.

Parameters:

<RightMargin> Right margin

Range: 0.01 mm to 10000 mm
*RST: n/a (*RST does not affect the printer configuration)

Example: `HCOP:PAGE:MARG:LEFT 10; RIGHT 10`
Set an left and a right margin of 1 cm.

HCOPy:PAGE:MARGin:TOP <UppMargin>

Defines the distance between the top of the page and the top of the printed information.

Parameters:

<UppMargin> Upper margin

Range: 0.01 mm to 10000 mm
*RST: n/a (*RST does not affect the printer configuration)

Example: `HCOP:PAGE:MARG:BOTT 10; TOP 10`
Set an upper and a lower margin of 1 cm.

HCOPy:PAGE:ORIentation <Orientation>

Defines the orientation of the printed page. Switching between `LANDscape` and `PORTrait` rotates the hardcopy result by 90 degrees. No other settings are changed.

Parameters:

<Orientation> LANDscape | PORTrait

LANDscape - long edge of the paper is the top of the page.
PORTrait - short edge of the paper is the top of the page.

*RST: n/a (*RST does not affect the printer configuration)

Example: `HCOP:PAGE:ORI LAND; :HCOP`
Select landscape page orientation and start printing.

HCOPy:PAGE:WINDow <PrintDiagram>

Defines how the diagrams shall be printed (using `HCOPy[:IMMediate]`.

Parameters:

<PrintDiagram> ALL | SINGle | ACTive | HARDcopy | NONE

ALL - all diagrams are printed on one page.
SINGle - one diagram per page.
ACTive - print only active diagram.
HARDcopy - print a screenshot of the diagram area, preserving layout and colors ("real screenshot")
NONE - print no diagram at all.

*RST: n/a (*RST does not affect the printer configuration)

Example: `HCOP:PAGE:WIND SING; :HCOP`
Select one diagram per page and start printing.

7.3.8 INITiate commands

The `INITiate...` commands control the initiation of the trigger system and define the scope of the triggered measurement.

INITiate<Ch>:CONTinuous....1247
INITiate:CONTinuous:ALL....1248
INITiate[:IMMediate]:ALL....1249
INITiate<Ch>[:IMMediate][:DUMMy]....1249
INITiate<Ch>[:IMMediate]:SCOPe....1250
INITiate:PIPelining:STATe....1250
INITiate<Ch>:STOP....1250

INITiate<Ch>:CONTinuous <Boolean>

Qualifies whether the analyzer measures in single sweep or in continuous sweep mode.

Suffix:

<Ch> Channel number.
This suffix is ignored in the "ZVR" and "ZVABT" compatibility modes (`SYSTem:LANGuage 'ZVR' | 'ZVABT'`).

Parameters:

<Boolean> ON - the analyzer measures continuously, repeating the current sweep.
OFF - the measurement is stopped after the number of sweeps defined via `[SENSe<Ch>:]SWEep:COUNt`. `INITiate<Ch>[:IMMediate][:DUMMy]` initiates a new measurement cycle.

*RST: ON

Example:

```
*RST; :CALC2:PAR:SDEF 'TRC2','S11'
DISPlay:WINDow:TRACe2:FEED 'Trc2'
```
Reset the analyzer to create the default channel no. 1 and default trace. Create a second trace and display the trace. Both traces are measured continuously.
```
INIT1:CONT OFF
```
Activate single sweep mode for the first channel. The measurement in channel no. 1 is stopped after the current sweep. Channel no. 2 is still measured continuously.
```
INIT1
```
Activate a new (single) sweep in channel no. 1. Channel no. 2 is still measured continuously.
```
INIT:CONT:ALL OFF
```
Activate single sweep mode for all channels. The measurement in channel no. 2 is also stopped after the current sweep.
```
INIT:ALL
```
Re-start a single sweep in both channels.

Example:

Alternative settings using the R&S ZVAB compatibility mode:
```
*RST; :SYSTem:LANGuage 'ZVABT'
*RST; :INIT:CONT OFF
```
Activate single sweep mode for all channels (including channel no. 2 created later).
```
INIT:SCOP SING
```
State that a single sweep will be performed in the active channel.
```
CALC2:PAR:SDEF 'TRC2','S11'; :INIT2
```
Create channel no. 2 with a new trace and start a single sweep in channel no. 2. Start a single sweep in the second channel.

Manual operation: See "Continuous / Single" on page 559

INITiate:CONTinuous:ALL <Boolean>

Qualifies whether the analyzer measures in single sweep or in continuous sweep mode.

This command is not supported in the "ZVR" or "ZVABT" compatibility modes.

Parameters:

<Boolean> ON - the analyzer measures continuously in all channel, repeating the current sweep. The query returns ON (1) if at least one channel is measured continuously.
OFF - the measurement is stopped after the number of sweeps defined via `[SENSe<Ch>:]SWEep:COUNt`. `INITiate<Ch>[:IMMediate][:DUMMy]` initiates a new measurement cycle.

*RST: ON

Example: See `INITiate<Ch>:CONTinuous`

Manual operation: See "All Channels Continuous/All Channels on Hold" on page 560

INITiate[:IMMediate]:ALL

Starts a new single sweep sequence in all channels. This command is available in single sweep mode only (`INITiate:CONTinuous:ALL OFF`). The data of the last sweep (or of previous sweeps, see Retrieving the results of previous sweeps) can be read using `CALCulate<Chn>:DATA:NSWeep:FIRSt? SDATa, <count>`.

This command is not supported in the "ZVR" or "ZVABT" compatibility modes.

Note: In contrast to all other commands of the analyzer, the `INITiate<Ch>[:IMMediate]...` commands have been implemented for overlapped execution; see Section 6.4, "Command processing", on page 886.

Example: See `INITiate<Ch>:CONTinuous`

Usage: Event

Manual operation: See "Restart Sweep" on page 560

INITiate<Ch>[:IMMediate][:DUMMy]

Starts a new single sweep sequence. This command is available in single sweep mode only (`INITiate<Ch>:CONTinuous OFF`). The data of the last sweep (or previous sweeps, see Section 8.2.4.3, "Retrieving the results of previous sweeps", on page 1670) can be read using `CALCulate<Chn>:DATA:NSWeep:FIRSt? SDATa, <count>`.

Note: In contrast to all other commands of the analyzer, the `INITiate<Ch>[:IMMediate]...` commands have been implemented for overlapped execution; see Section 6.4, "Command processing", on page 886.

Suffix:

<Ch> Channel number. If the channel number does not exist the analyzer returns an error message. If the "ZVR" or "ZVABT" compatibility mode is active and `INITiate<Ch>[:IMMediate]:SCOPe ALL` is selected, this suffix is ignored.

Example: See `INITiate<Ch>:CONTinuous`

Usage: Event

Manual operation: See "Restart Sweep" on page 560

INITiate<Ch>[:IMMediate]:SCOPe <Scope>

Selects the scope of the single sweep sequence. The setting is applied in single sweep mode only (`INITiate<Ch>:CONTinuous OFF`).

This command is required in compatibility modes only (see `SYSTem:LANGuage`).

Suffix:
<Ch> Channel number

Parameters:
<Scope> ALL | SINGle

`INITiate<Ch>[:IMMediate][:DUMMy]` starts a single sweep in all channels or in the referenced channel `<Ch>` only.

Example: See `INITiate<Ch>:CONTinuous`

Manual operation: See "Sweep All Channels/Sweeps" on page 562

INITiate:PIPelining:STATe <Mode>

Allows you to activate pipelining of measurement data.

This applies to all channels in the active setup.

Parameters:
<Mode> OFF | CHANnel

OFF
No pipelining (default)

CHANnel
Pipeline enabled

Manual operation: See "Pipelining" on page 561

INITiate<Ch>:STOP

Stops an ongoing sweep in channel <Ch>

Suffix:
<Ch> Channel number

Usage: Event

Manual operation: See "Stop Sweep" on page 560

7.3.9 INSTrument commands

The `INSTrument...` commands select or query particular resources (SCPI: logical instruments) of the analyzer.

INSTrument:NSELect 1251
INSTrument:PORT:COUNt? 1251
INSTrument:SMATrix 1251
INSTrument:TPORt:COUNt? 1252

INSTrument:NSELect <Channel>

Selects a channel as the active channel.

Parameters:

<Channel> Number of the channel to be activated. The channel must be created before using `CONFigure:CHANnel<Ch>[:STATe] ON`.

Range: 1, 2, ...
*RST: 1

Example: `CONF:CHAN2:STAT ON; :INST:NSEL?`
Create channel no. 2 and select it as the active channel. The query returns 2.

INSTrument:PORT:COUNt?

Returns the number of test ports (Port 1, Port 2 ...) of the analyzer.

Example: `INST:PORT:COUN?`
Return the number of ports of your analyzer.

Usage: Query only

INSTrument:SMATrix <Boolean>

In "set direction" this command perfoms a default setup or "clear" of the switch matrix RF connections; in "get direction" it queries whether at least one RF connection to a switch matrix is configured.

Parameters:

<Boolean> ON | 1 (setting) – adds all registered switch matrices to the RF configuration, performing a default assignment of VNA ports and test ports.
OFF | 0 (setting) – removes all switch matrices from the RF configuration
1 (query) – at least one RF connection to a switch matrix
0 (query) – no RF connection to a switch matrix

Example: See `SYSTem:COMMunicate:RDEVice:SMATrix:CONFigure:STARt`

INSTrument:TPORt:COUNt?

Returns the total number of test ports.

In absence of switch matrices, i.e. if no RF connection to a switch matrix is configured, this is identical to the number of VNA ports (see `INSTrument:PORT:COUNt?`).With switch matrices, the return value is the total number of matrix test ports plus the number of VNA ports that are not connected to a switch matrix.

Usage: Query only

7.3.10 MEMory commands

The `MEMory...` commands control the loaded recall sets of the analyzer.



Storing setups

The `MEMory...` commands do not affect any stored files. Use the `MMEMory...` commands to store and load data and to manage files stored on a mass storage device.

MEMory:CATalog? 1252
MEMory:CATalog:COUNt? 1252
MEMory:DEFine 1253
MEMory:DELete:ALL 1253
MEMory:DELete[:NAME] 1253
MEMory:SELect 1253

MEMory:CATalog?

Returns the names of all loaded recall sets.

Example:
```
*RST; :MEM:DEF 'SET_2'
```
Create a recall set named 'Set_2' and make it the active recall set.
```
MEM:CAT?
```
Query all recall sets. The response is `'Set1,SET_2'`.
```
MMEM:STOR:STAT 1, 'C:\Users\Public\
Documents\Rohde-Schwarz\Vna\RecallSets\Set_2.znx'
MEM:DEL 'Set_2.znx'
```
Store the active recall set "Set_2" to a file, renaming it "Set_2.znx". Close the setup.

Usage: Query only

MEMory:CATalog:COUNt?

Returns the number of loaded recall sets.

Usage: Query only

MEMory:DEFine <Name>

Creates a new recall set<Name> using default settings for the traces, channels and diagram areas. The created recall set becomes the active recall set.

Setting parameters:

<Name> String parameter to specify the name of the created recall set.

Example: See `MEMory:CATalog?`

Usage: Setting only

Manual operation: See "New" on page 780

MEMory:DELete:ALL

Deletes all loaded recall sets.

Example: `MEM:DEL:ALL; :MEM:CAT?`
Delete all recall sets. The query `MEM:CAT?` returns an empty string. The local screen shows no recall set.

Usage: Event

MEMory:DELete[:NAME] <Name>

Closes the specified recall set.

Setting parameters:

<Name> String parameter to specify the name of the recall set to be closed.

Example: See `MEMory:CATalog?`

Usage: Setting only

MEMory:SELect <RecallSet>

Selects a recall set as the active recall set or returns the name of the active recall set.

Parameters:

<RecallSet> String parameter to specify the recall set.

Example:
```
*RST; :MEM:DEF 'SET_2'
```
Create a recall set named "SET_2" and make it the active recall set.
```
MEM:SEL 'Set1'
```
Activate the default recall set"Set1".
```
MMEM:STOR:STAT 1, 'C:\Users\Public\
Documents\Rohde-Schwarz\Vna\RecallSets\
Set1.znx'; :MEM:DEL 'Set1.znx'
```
Store the active recall set"Set1" to a file, renaming it `Set1.znx`. Close the recall set.

7.3.11 MMEMory commands

The `MMEMory...` commands provide mass storage capabilities for the analyzer.

Internal and external mass storage

The mass storage of the analyzer can be internal or external. The internal mass storage location is either the public folder or the instrument folder of the internal hard disk (`C:\Users\Public` or `C:\Users\Instrument`, see below). The external mass storage device can be a USB memory stick connected to one of the USB ports (mapped to any free drive letter) or a network connection.

File and directory names

The parameters for file names and directory names are strings. Some commands use a fixed "working" directory. For others, the file name parameter must contain the absolute path including the drive name and all subdirectories. If the specified path is not absolute, the file location is interpreted relative to the current directory (queried with `MMEMory:CDIRectory`). The file name itself can contain the period as a separator for extensions.

File and directory names can be chosen according to Windows® conventions. All letters and numbers are allowed, plus the special characters "_", "^", "$", "~", "!", "#", "%", "&", "-", "{", "}", "(", ")", "@" and "`". Reserved file names are CON, AUX, COM1, ..., COM4, LPT1, ..., LPT3, NUL and PRN. The use of wildcards ? and * is not allowed.

Public folders in Windows® and default file locations

To achieve maximum system security, most of the folders on the internal hard disk are read-only folders. Only the following folders can be changed:

- `C:\Users\Public`
- `C:\Users\Instrument`

The public and instrument folders can be used to store user data. To simplify this task, the public folder contains predefined subfolders; e.g. the subfolder `C:\Users\Public\Documents\Rohde-Schwarz\Vna\Calibration\Kits` is intended for calibration kit data. The subfolder structure is similar to R&S ZVA/B instruments, however, the R&S ZVA/B default path `C:\Rohde&Schwarz\Nwa` is replaced by `C:\Users\Public\Documents\Rohde-Schwarz\Vna..`

MMEMory:AKAL:FACTory:CONVersion....1256
MMEMory:AKAL:USER:CONVersion....1256
MMEMory:CATalog?....1257
MMEMory:CATalog:ALL?....1258
MMEMory:CDIRectory....1258
MMEMory:CKIT:INFO?....1259
MMEMory:COPY....1259
MMEMory:DATA....1259
MMEMory:DELete....1260
MMEMory:DELete:CORRection....1260
MMEMory:FAVorite<FavId>....1260
MMEMory:LOAD:CABLe....1261

MMEMory:LOAD:CKIT ... 1261
MMEMory:LOAD:CKIT:SDATa ... 1262
MMEMory:LOAD:CKIT:SDATa:WLABel ... 1263
MMEMory:LOAD:CKIT:UDIRectory ... 1264
MMEMory:LOAD:CMAP ... 1265
MMEMory:LOAD:CMAP:HCOPy ... 1265
MMEMory:LOAD:CORRection ... 1266
MMEMory:LOAD:CORRection:MERGe ... 1266
MMEMory:LOAD:CORRection:RESolve ... 1267
MMEMory:LOAD:CORRection:TCOefficient<Ch> ... 1268
MMEMory:LOAD:EYE:BPATtern ... 1269
MMEMory:LOAD:EYE:EQUalization:CTLE ... 1269
MMEMory:LOAD:EYE:EQUalization:DFE ... 1270
MMEMory:LOAD:EYE:EQUalization:FFE ... 1270
MMEMory:LOAD:EYE:JITTer ... 1270
MMEMory:LOAD:EYE:MASK ... 1271
MMEMory:LOAD:LIMit ... 1271
MMEMory:LOAD:RIPPle ... 1272
MMEMory:LOAD:SEGMent ... 1273
MMEMory:LOAD:STATe ... 1274
MMEMory:LOAD:TRACe ... 1274
MMEMory:LOAD:TRACe:AUTO ... 1275
MMEMory:LOAD:VNETworks<Ch>:BALanced:EMBedding<LogPt> ... 1276
MMEMory:LOAD:VNETworks<Ch>:BALanced:DEEMbedding<LogPt> ... 1276
MMEMory:LOAD:VNETworks<Ch>:DIFFerential:EMBedding<LogPt> ... 1277
MMEMory:LOAD:VNETworks<Ch>:GLOop:DEEMbedding<group> ... 1278
MMEMory:LOAD:VNETworks<Ch>:GLOop:EMBedding<group> ... 1278
MMEMory:LOAD:VNETworks<Ch>:PPAir:EMBedding<ListId> ... 1279
MMEMory:LOAD:VNETworks<Ch>:PPAir:DEEMbedding<ListId> ... 1279
MMEMory:LOAD:VNETworks<Ch>:SENDed:EMBedding<PhyPt> ... 1280
MMEMory:LOAD:VNETworks<Ch>:SENDed:DEEMbedding<PhyPt> ... 1280
MMEMory:MDIRectory ... 1281
MMEMory:MOVE ... 1282
MMEMory:MSIS ... 1282
MMEMory:NAME ... 1282
MMEMory:RDIRectory ... 1283
MMEMory:SIMulation:DELete:ALL ... 1283
MMEMory:SIMulation:LOAD ... 1283
MMEMory:SIMulation:SDATa<Idx> ... 1284
MMEMory:SIMulation:SDATa<Idx>:DELete ... 1284
MMEMory:SIMulation:SDATa<Idx>:LOAD ... 1284
MMEMory:STORe:CABLe ... 1285
MMEMory:STORe:CKIT ... 1285
MMEMory:STORe:CKIT:WLABel ... 1286
MMEMory:STORe:CMAP ... 1286
MMEMory:STORe:CMAP:HCOPy ... 1286
MMEMory:STORe:CORRection ... 1287
MMEMory:STORe:CORRection:TCOefficient<Ch> ... 1287
MMEMory:STORe:EYE:EQUalization:CTLE ... 1288
MMEMory:STORe:EYE:EQUalization:DFE ... 1288

MMEMory:STORe:EYE:EQUalization:FFE....1289
MMEMory:STORe:EYE:MASK....1289
MMEMory:STORe:EYE:MASK:RESults....1289
MMEMory:STORe:EYE:MEASurements....1290
MMEMory:STORe:LIMit....1290
MMEMory:STORe:MARKer....1291
MMEMory:STORe:RIPPle....1291
MMEMory:STORe:SEGMent....1292
MMEMory:STORe:STATe....1292
MMEMory:STORe:TRACe....1292
MMEMory:STORe:TRACe:CHANnel....1294
MMEMory:STORe:TRACe:OPTion:BALanced....1295
MMEMory:STORe:TRACe:OPTion:SYMMetric....1295
MMEMory:STORe:TRACe:OPTion:COMMent....1295
MMEMory:STORe:TRACe:OPTion:DECimals:DATA....1296
MMEMory:STORe:TRACe:OPTion:DECimals:STIMulus....1296
MMEMory:STORe:TRACe:OPTion:EXTRapolate....1296
MMEMory:STORe:TRACe:OPTion:FORMat....1296
MMEMory:STORe:TRACe:OPTion:PLUS....1297
MMEMory:STORe:TRACe:OPTion:RSAMple....1297
MMEMory:STORe:TRACe:OPTion:RSAMple:POINts....1297
MMEMory:STORe:TRACe:OPTion:RSAMple:PRESet....1298
MMEMory:STORe:TRACe:OPTion:RSAMple:TIMStep....1298
MMEMory:STORe:TRACe:OPTion:SSEParator....1298
MMEMory:STORe:TRACe:OPTion:TABS....1298
MMEMory:STORe:TRACe:OPTion:TRIM....1299
MMEMory:STORe:TRACe:PORTs....1299

MMEMory:AKAL:FACTory:CONVersion <Directory>

Converts the factory calibration data of the standards in the active calibration unit (`SYSTem:COMMunicate:RDEVice:AKAL:ADDRess`) to Touchstone format and copies it to the specified directory.

Setting parameters:

<Directory> String parameter to specify the directory.

Example: `MMEM:AKAL:FACTory:CONVersion 'C:\Users\Public\Documents\Rohde-Schwarz\Vna\AKAL\Touchstone'`
Convert and copy the factory calibration data of the active calibration unit to the specified (writable) directory.

Usage: Setting only

Manual operation: See "Apply/Cancel" on page 576

MMEMory:AKAL:USER:CONVersion <Directory>[, <CalKitFile>]

Converts an arbitrary (e.g. user-defined) set of calibration data of the standards in the active calibration unit (`SYSTem:COMMunicate:RDEVice:AKAL:ADDRess`) to Touchstone format and copies it to the specified directory.

Setting parameters:

<Directory> String parameter to specify the directory.

<CalKitFile> Name and (possibly) directory of the cal kit file to be used for the automatic calibration (optional string parameter):
– If the parameter is omitted, the analyzer uses the last characterized cal kit file.
– If an empty string (`' '`) is specified, the factory cal kit file stored in the active calibration unit (`SYSTem:COMMunicate:RDEVice:AKAL:ADDRess`) is used. By default this file is also used in manual control.
– A cal kit file name `*.calkit` without path denotes a specific cal kit file stored in the active calibration unit.
– A cal kit file name `*.calkit` with path denotes a specific cal kit file stored in an arbitrary directory.

Example: `MMEM:AKAL:USER:CONV 'C:\Users\Public\Documents\Rohde-Schwarz\Vna\AKAL\Touchstone'`
Convert and copy the calibration data of the standards of the last characterized cal kit to the specified (writable) directory.
`MMEM:AKAL:USER:CONV 'C:\Users\Public\Documents\Rohde-Schwarz\Vna\AKAL\Touchstone', 'user.calkit'`
Convert and copy the calibration data of the standards of the user-defined cal kit `'user.calkit'` to the specified directory.

Usage: Setting only

Manual operation: See "Apply/Cancel" on page 576

MMEMory:CATalog? [<Directory>]

Returns the contents of the current or of a specified directory. The directory information is returned in the following format:

<Used Size>, <FreeDiskSpace>{,<FileEntry>}{,<DirectoryEntry>}

The information elements indicate the following:

- <Used Size> – disk space in bytes used by the listed files, excluding subdirectories
- <FreeDiskSpace> – available free disk space in bytes
- <FileEntry> – file name, (blank), file size in bytes
- <DirectoryEntry> – directory name, <Dir>, (blank)

Tip: Use `MMEMory:CATalog:ALL?` to query the contents of the current directory and all subdirectories.

Query parameters:

<Directory> String parameter to specify the directory. If the directory is omitted, the command queries the contents of the current directory, to be queried with `MMEMory:CDIRectory?`.

Example: `MMEM:CAT?`
Possible response: `0, 771604480, calibration, <DIR>, , colorschemes, <DIR>, ,....`

Usage: Query only

MMEMory:CATalog:ALL? [<Directory>]

Returns the contents of the current or of a specified directory and all subdirectories. The information is returned in the following format:

Directory of <Directory>, <Used Size>, <FreeDiskSpace>{,<FileEntry>}{,<DirectoryEntry>}{,Directory of <Subdirectory>, <Used Size>, <FreeDiskSpace>{,<FileEntry>}{,<DirectoryEntry>}}

See also `MMEMory:CATalog?`.

Tip: Use `MMEMory:CATalog?` to query the contents of the current directory.

Query parameters:

<Directory> String parameter to specify the directory path. If the directory is omitted, the command queries the contents of the current directory, to be queried with `MMEMory:CDIRectory?`

Example: `MMEM:CAT:ALL?`
Possible response:
`Directory of C:\Users\Public\Documents\Rohde-Schwarz\Vna\,0, 2283155456, calibration, <DIR>, , colorschemes, <DIR>, , hardcopy, <DIR>, , limitlines, <DIR>, , ...`

Usage: Query only

MMEMory:CDIRectory <Directory>

Changes the *current directory* for `MMEMory` commands.

Relative paths are interpreted relative to this directory.

Parameters:

<Directory> String parameter to specify the directory. If `DEFault` is used, the analyzer selects the default directory `C:\Users\Public\Documents\Rohde-Schwarz\Vna`.

Example: `MMEMory:CDIRectory 'C:\temp'`
sets the current directory to `C:\temp`.
`MMEMory:CATalog? 'somedir'`
lists the content of `C:\temp\somedir`.
See also `MMEMory:MSIS` and the condensed programming example in section Path-independent remote control programs.

MMEMory:CKIT:INFO? <CalKitFile>[, <Detail>]

Queries connector type, name, label and gender of a cal kit defined in the specified cal kit file

Query parameters:

<CalKitFile> Path to the cal kit file, either absolute or relative to the current directory (see `MMEMory:CDIRectory`)

<Detail> CONNector | LABel | NAME | GENDer

If specified, the command only returns the corresponding property

Return values:

<ConnectorType>

<CalKitName>

<KitLabel>

<Gender> **1**: has a gender
0: doesn't have a gender

Usage: Query only

MMEMory:COPY <SourceFile>, <NewFile>

Copies an existing file to a new file.

Setting parameters:

<SourceFile>

<NewFile> String parameters to specify the name of the file to be copied and the name of the new file.

*RST: n/a

Example:

```
MMEM:COPY 'C:\Users\Public\Documents\
Rohde-Schwarz\Vna\RecallSets\SET1.znx','D:'
```

Copy file `Set1.znx` in directory `C:\Users\Public\Documents\Rohde-Schwarz\Vna\RecallSets` to the external storage medium, mapped to drive `D:`.

Usage: Setting only

MMEMory:DATA <Filename>[, <DataBlock>]

Loads a `<DataBlock>` into the file `<FileName>`.

Parameters:

<Filename> String parameter to specify the name of the file.

<DataBlock> <block_data>

Data in IEEE488.2 block data format. The delimiter EOI must be selected to achieve correct data transfer.

Example: `MMEM:DATA? 'C:\Users\Public\TEST01.HCP'`
Query the block data contained in file `TEST01.HCP`.

MMEMory:DELete <File>[, <Force>]

Removes a file from the specified directory.

Setting parameters:

<File> Mandatory string parameter containing the path and file name of the removed file. If the path is omitted, the current directory is used (see `MMEMory:CDIRectory`).

<Force> FORCe

Optional parameter, allows you to delete read-only files, too.

Example: `MMEM:DEL 'C:\Users\Public\TEST01.HCP'`
Remove file `TEST01.HCP` from the directory `C:\Users\Public`. The file must not be read-only; otherwise the additional parameter `FORCe` is required.

Usage: Setting only

MMEMory:DELete:CORRection <CalGroupName>

Deletes a system error correction data set stored in the cal pool (cal group file).

Setting parameters:

<CalGroupName> String parameter to specify the name of the cal group file to be deleted. Cal group files must have the extension `*.cal`. The directory path must not be specified; the analyzer always uses the default cal pool directory `C:\Users\Public\Documents\Rohde-Schwarz\Vna\Calibration\Data`.

Example: See `MMEMory:LOAD:CORRection`

Usage: Setting only

Manual operation: See "Pool / Delete from Pool" on page 659

MMEMory:FAVorite<FavId> <RecallSetFile>

Manages the list of favorite recall sets

Suffix:

<FavId> Position in the favorites list

Setting parameters:

<RecallSetFile> File path, either absolute or relative to the current directory (see `MMEMory:CDIRectory`). The empty string represents an empty position in the favorites list.
Note that when a non-empty favorite is set, the target recall set must exist.

Example:
```
:MMEMory:CDIRectory DEFault
:MMEM:FAV1 'RecallSets\My_RecallSet1.znx'
```
Sets `My_RecallSet1.znx` at position 1 of the favorites list.
```
MMEM:FAV1?
```
Returns `'C:\Users\Public\Documents\Rohde-Schwarz\Vna\RecallSets\My_RecallSet1.znx'`
```
MMEM:FAV1 'RecallSets\My_RecallSet2.znx'
```
Sets `My_RecallSet2.znx` as favorite 1, overwriting the previous favorite 1
```
MMEM:FAV1 ''
```
Clears position 1 of the favorites list
```
MMEM:FAV1?
```
Returns `''`

Manual operation: See "Import" on page 783

MMEMory:LOAD:CABLe <InputDir>

Loads Distance to Fault (DtF) cable type files (`*.rsc`) from the specified input directory.

Cable types are loaded "by name", overwriting user-defined cable types of the same name. Predefined cable types of the same name will not be overwritten.

Setting parameters:

<InputDir> Input Directory

Example:
```
MMEMory:STORe:CABLe 'C:\dtf-cables'
```
Saves all predefined and user-defined Distance to Fault (DtF) cable types to the directory `C:\dtf-cables`.
Edit `*.rsc` files for user defined cable types, copy `*.rsc` files, rename and edit them. Afterwards execute
```
MMEMory:LOAD:CABLe 'C:\dtf-cables'
```
to update existing and create new user-defined cable types.

Usage: Setting only

Options: R&S ZNB-K3

Manual operation: See "Add / Delete" on page 372

MMEMory:LOAD:CKIT <CalKitFile>

Loads cal kit data from the specified cal kit file.

Setting parameters:

<CalKitFile> String parameter to specify name and directory of the loaded cal kit file. If no path is specified the analyzer searches the current directory, to be queried with `MMEMory:CDIRectory`

Example: `MMEM:LOAD:CKIT 'C:\Users\Public\Documents\Rohde-Schwarz\Vna\Calibration\Kits\New_kit.calkit'`
Load the previously created cal kit file `New_kit.calkit` from the default cal kit directory.
`... :MMEM:STOR:CKIT 'New_kit', 'C:\Users\Public\Documents\Rohde-Schwarz\Vna\Calibration\Kits\New_kit.calkit'`
Store the data for the user-defined cal kit `Newkit` and overwrite the cal kit file `New_kit.calkit`.

Usage: Setting only

Manual operation: See "Import Cal Kit" on page 587

MMEMory:LOAD:CKIT:SDATa <ConnectorType>, <CalKitName>, <StandardType>, <StandardLabel>, <TouchstoneFile>[, <FirstPort>[, <SecondPort>]]

Loads cal kit data for a calibration standard from a specified Touchstone file. A restriction on the port assignment may be defined in addition.

Use the newer command `MMEMory:LOAD:CKIT:SDATa:WLABel` to be able to distinguish cal kits by label.

Setting parameters:

<ConnectorType> String parameter containing the name of the connector type.

<CalKitName> String parameter containing the name of a calibration kit available on the analyzer.

<StandardType> MMTHrough | MFTHrough | FFTHrough | MMLine | MMLine1 | MMLine2 | MMLine3 | MMLine4 | MMLine5 | MMLine6 | MFLine | MFLine1 | MFLine2 | MFLine3 | MFLine4 | MFLine5 | MFLine6 | FFLine | FFLine1 | FFLine2 | FFLine3 | FFLine4 | FFLine5 | FFLine6 | MMATten | MFATten | FFATten | MMSNetwork | MFSNetwork | FFSNetwork | MOPen | FOPen | MSHort | FSHort | MOSHort | MOSHort1 | MOSHort2 | MOSHort3 | FOSHort | FOSHort1 | FOSHort2 | FOSHort3 | MREFlect | FREFlect | MMTCh | FMTCh | MSMatch | FSMatch

Standard types; for a description refer to table Standard types and their parameters.

<StandardLabel> String parameter addressing a particular calibration standard by its label. An empty string means that no label is defined.

<TouchstoneFile> String parameter to specify name and directory of the Touchstone file to be loaded. A `*.s1p` file must be used for one-port standards, a `*.s2p` file for two-port standards.
If no path is specified the analyzer searches the current directory, to be queried with `MMEMory:CDIRectory`

*RST: -

<FirstPort> First port number (sufficient for one-port standards). If the port numbers are omitted, the cal kit data is valid for all ports.

<SecondPort> Second port number, for two-port standards. If the port numbers are omitted, the cal kit data is valid for all ports.

Example:
```
MMEM:LOAD:CKIT:SDAT 'N 50 Ohm','Default Kit',MOPEN,'Test data','test.s1p',1
```
Load the file Test.s1p from the current directory in order to define the properties of an Open (m) standard in the cal kit named "Default Kit" for the N 50 Ω connector type. Assign the label "Test data" and specify that the standard data is valid for port no. 1 only.

Usage: Setting only

Manual operation: See "Read .s1p File / Read .s2p File /" on page 641

Connector and cal kit naming conventions:

Connector and calibration kit names must be entered as string parameters. The strings contain the connector and cal kit names used in the `Calibration Kits` dialog; a Ω in the name must be replaced by 'Ohm', e.g.:

– `'NewKit1'` denotes the user-defined calibration kit "NewKit1".

– `'N 50 Ohm Ideal Kit'` denotes the "N 50 Ω Ideal Kit".

– `'ZV-Z21 typical'` denotes the cal kit "ZV-Z21 typical".

MMEMory:LOAD:CKIT:SDATa:WLABel <ConnectorType>, <CalKitName>, <KitLabel>, <StandardType>, <StandardLabel>, <TouchstoneFile>[, <FirstPort>[, <SecondPort>]]

Loads characterization data from the given Touchstone file; similar to existing command `MMEMory:LOAD:CKIT:SDATa` but supports cal kit addressing **by label**.

Setting parameters:

<ConnectorType> String parameter containing the name of the connector type.

<CalKitName> String parameter containing the name of a calibration kit available on the analyzer.

<KitLabel> String parameter containing the label of a calibration kit available on the analyzer. An empty string means that no label is defined.

<StandardType> MMTHrough | MFTHrough | FFTHrough | MMLine | MMLine1 | MMLine2 | MMLine3 | MMLine4 | MMLine5 | MMLine6 | MFLine | MFLine1 | MFLine2 | MFLine3 | MFLine4 | MFLine5 | MFLine6 | FFLine | FFLine1 | FFLine2 | FFLine3 | FFLine4 | FFLine5 | FFLine6 | MMATten | MFATten | FFATten | MMSNetwork | MFSNetwork | FFSNetwork | MOPen | FOPen | MSHort | FSHort | MOSHort | MOSHort1 | MOSHort2 | MOSHort3 | FOSHort | FOSHort1 | FOSHort2 | FOSHort3 | MREFlect | FREFlect | MMTCh | FMTCh | MSMatch | FSMatch

Standard types; for a description refer to table Standard types and their parameters.

<StandardLabel> String parameter addressing a particular calibration standard by its label. An empty string means that no label is defined.

<TouchstoneFile> String parameter to specify the name and directory of the Touchstone file to be loaded. A `*.s1p` file must be used for one-port standards, a `*.s2p` file for two-port standards.
If no path is specified the analyzer searches the current directory, to be queried with `MMEMory:CDIRectory`

<FirstPort> First port number (sufficient for one-port standards). If the port numbers are omitted, the cal kit data is valid for all ports.

<SecondPort> Second port number, for two-port standards. If the port numbers are omitted, the cal kit data is valid for all ports.

Example:
```
MMEM:LOAD:CKIT:SDAT:WLAB 'N 50 Ohm','Default Kit','0815',MOPEN,'Test data','test.s1p',1
```
Load the file Test.s1p from the current directory in order to define the properties of an Open (m) standard in the cal kit "Default Kit" with label "0815" for the N 50 Ω connector type. Assign the label "Test data" and specify that the standard data is valid for port no. 1 only.

Usage: Setting only

Connector and cal kit naming conventions:

Connector and calibration kit names must be entered as string parameters. The strings contain the connector and cal kit names used in the `Calibration Kits` dialog; a Ω in the name must be replaced by 'Ohm', e.g.:

– `'NewKit1'` denotes the user-defined calibration kit "NewKit1".

– `'N 50 Ohm Ideal Kit'` denotes the "N 50 Ω Ideal Kit".

– `'ZV-Z21 typical'` denotes the cal kit "ZV-Z21 typical".

MMEMory:LOAD:CKIT:UDIRectory <Directory>

Specifies the "Search Path for Additional Cal Kits and Connector Types". All cal kit files in the specified directory will be (re-)loaded automatically as predefined kits (i.e. read-only kits which cannot be modified) every time the VNA application is started.

Parameters:

<Directory> String parameter to specify the directory path. The directory can be created separately (`MMEMory:MDIRectory`). An empty string means that no cal kit files will be loaded.

Example:

```
MMEM:LOAD:CKIT:UDIR 'C:\Users\Public\Documents
\Rohde-Schwarz\Vna\Calibration\Kits\Autoload'
```
Specifiy the directory for additionally available cal kits.
```
MMEM:MDIR 'C:\Users\Public\Documents
\Rohde-Schwarz\Vna\Calibration\Kits\Autoload'
```
Create the specified "Autoload" directory.
```
MMEM:STOR:CKIT 'New Kit 1', 'C:
\Users\Public\Documents
\Rohde-Schwarz\Vna\Calibration\Kits\Autoload
\New Kit 1.calkit'
```
Store the data for the existing, user-defined cal kit "New Kit 1" to the "Autoload" directory.

Manual operation: See "Search Path for additional Cal Kits and Connector Types" on page 814

MMEMory:LOAD:CMAP <ColorSchemeFile>
MMEMory:LOAD:CMAP:HCOPy <ColorSchemeFile>

Loads a color scheme from a the VNA color scheme file and activates it as the user-defined color scheme for display or printing (`HCOPy`).

Setting parameters:

<ColorSchemeFile> String parameter to specify the name and directory of the color scheme file to be loaded. The default extension (manual control) for color scheme files is `*.ColorScheme`, although other extensions are allowed.

Example:

```
MMEM:LOAD:CMAP 'C:\Users\Public
\Documents\Rohde-Schwarz\Vna\ColorSchemes\
Test.ColorScheme'
```
Load the previously created color scheme file `Test.ColorScheme` from the default color scheme directory.
```
DISP:CMAP13:RGB 1,0,0; :DISP:CMAP14:RGB 0,1,0
```
Color the first trace red, the second trace green.
```
MMEM:STOR:CMAP 'C:\Users\Public
\Documents\Rohde-Schwarz\Vna\ColorSchemes\
Test.ColorScheme'
```
Store the data for the user-defined cal kit "Newkit" and overwrite the cal kit file `New_kit.calkit`.

Usage: Setting only

Manual operation: See "Recall... / Save..." on page 806

MMEMory:LOAD:CORRection <Channel>[, <CalGroupFile>]

Applies a system error correction data set stored in the cal pool (cal group file) to channel `<Channel>`.

Parameters:

<Channel> Channel number of an existing channel. `ALL` applies the selected data set to all channels.

<CalGroupFile> String parameter to specify the name of the cal group file to be loaded. Cal group files must have the extension `*.cal`. The directory path must not be specified; the analyzer always uses the default cal pool directory `C:\Users\Public\Documents\Rohde-Schwarz\Vna\Calibration\Data`.

Example:

```
MMEM:STOR:CORR 1,'Calgroup1.cal'
```
Copy the current correction data set of channel 1 to a cal group file `Calgroup1.cal`.
```
CONF:CHAN2:STAT ON; :MMEM:LOAD:CORR 2,
'Calgroup1.cal'
```
Apply the stored correction data (cal group file) to channel 2.
```
MMEM:LOAD:CORR? 2
```
Query the cal group file for channel 2. Response:
```
'Calgroup1.cal'
MMEM:LOAD:CORR:RES 2,'Calgroup1.cal'
```
Resolve the pool link between channel 2 and the cal group file.
```
MMEM:LOAD:CORR? 2
```
Query the cal group file for channel 2. Response: `''`
```
MMEM:DEL:CORR 'Calgroup1.cal'
```
Delete the created cal group file.

Manual operation: See "Add / Add All / Replace / Apply / Apply to All" on page 658

MMEMory:LOAD:CORRection:MERGe <Channel>[, <CalGroupFile>[, <CalGroupFile>]...]

Merges (activates) several cal group files for channel no. <Channel> so that the query `[SENSe<Ch>:]CORRection:COLLect:METHod:DEFine?` returns a list of all merged calibration types (equivalent to the calibration pool list in the "Calibration Manager" dialog). The merged cal group files can be stored to a common file (see example).

Note that the calibrations to be merged must be based on the same frequency grid (identical frequency sweep points).

Setting parameters:

<Channel> Channel number of an existing channel

<CalGroupFile>

<CalGroupFile> String parameters with the names of the merged cal group files. Cal group files must have the extension `*.cal`. The file extensions must be specified as part of the string parameters. In contrast the directory path must not be specified; the analyzer always uses the default path `C:\Users\Public\Documents\Rohde-Schwarz\Vna\CALibration\DATA`.

Example: `SENS1:CORR:COLL:METH:DEF 'Test', FRTRans, 1, 3`
Select a bidirectional transmission normalization between ports 1 and 3 as a calibration type for channel 1.
`CORR:COLL:SAVE:SEL:DEF; :MMEM:STOR:CORR 1, 'P1-P3.cal'`
Create a default calibration data set for the selected calibration type and store the data to a cal group file.
`SENS1:CORR:COLL:METH:DEF 'Test', FRTRans, 1, 4`
Select a bidirectional transmission normalization between ports 1 and 4 as a calibration type for channel 1.
`CORR:COLL:SAVE:SEL:DEF; :MMEM:STOR:CORR 1, 'P1-P4.cal'`
Create a default calibration data set for the selected calibration type and store the data to a cal group file.
`CORR:COLL:METH:DEF?`
Query the active calibrations for channel 1. The response is `FRTR0104` (the last data set stored).
`MMEM:LOAD:CORR:`
`MERGE 1, 'P1-P3.cal', 'P1-P4.cal'; :CORR:COLL:METH:DEF?`
Merge the two calibration types and query the active calibrations again. The response is `FRTR0103,FRTR0104`.
`MMEM:STOR:CORR 1, 'Merged.cal'`
Store both sets of calibration data to a common cal group file.

Usage: Setting only

Manual operation: See "Add / Add All / Replace / Apply / Apply to All" on page 658

MMEMory:LOAD:CORRection:RESolve <Channel>[, <CalGroupFile>]

Resolves the pool link between channel `<Channel>` and a correction data set (cal group file). After resolving the pool link, the analyzer keeps the previous system error correction as a channel calibration ("Channel Cal"). A new calibration will replace the channel calibration but not overwrite the old cal group file (and not affect other channels).

Setting parameters:

<Channel> Channel number of an existing channel. `ALL` resolves the pool link for all channels.

<CalGroupFile> Optional string parameter to specify the name of the cal group file. Cal group files must have the extension `*.cal`. The directory must not be specified; the analyzer always uses the default path `C:\Users\Public\Documents\Rohde-Schwarz\Vna\Calibration\Data`. If there is no link between `<Ch>` and the specified file, the command has no effect. If no file is specified, the command resolves any link between `<Ch>` and an arbitrary cal group file.

Example: See `MMEMory:LOAD:CORRection`

Usage: Setting only

Manual operation: See "Resolve Pool Link / Remove Pool Link" on page 659

MMEMory:LOAD:CORRection:TCOefficient<Ch> <TraceFile>[, <Trace>[, <DeEmbeddingPosition>]]

Loads two-port transmission coefficients from the specified power meter correction file or trace file to channel <Ch>.

Suffix:

<Ch> Channel number

Setting parameters:

<TraceFile> String parameter specifying the name and directory of the loaded trace file. The R&S ZNB supports power meter correction list files (`*.pmcl`, generated using `MMEMory:STORe:CORRection:TCOefficient<Ch>`), `*csv`, and Touchstone (`*.s1p, *.s2p, ...`) files.
If no path is specified the analyzer searches the current directory, to be queried with `MMEMory:CDIRectory`
The file extensions `*.s<n>p`, `*csv`, and `*.pmcl` for Touchstone, ASCII, and power meter correction list files are mandatory.

*RST: n/a

<Trace> Optional string parameter: For multiport Touchstone files (`*.snp, n > 1`), the parameter refers to a particular S-parameter trace (`'S11', 'S12', ...`). For ASCII (`*.csv`) files, the parameter references a trace name in the file (case sensitive). If the parameter is omitted, the first trace in the specified file is imported.
This parameter is not used for power meter correction list files (`*.pmcl`).

*RST: n/a

<DeEmbeddingPosition> BOTH | DUT | PM

Determines the two-port the command refers to:

DUT
The two-port between VNA and DUT (during measurement)

PM
The two-port between VNA and PM (during power calibration)

BOTH
Both positions

(parameter omitted)
If the parameter is omitted, the command refers to the two-port selected using `SOURce<Ch>:POWer<PhyPt>:CORRection:TCOefficient:CALibration`.

Example: See `SOURce:POWer:CORRection:TCOefficient[:STATe]` on page 1508

Usage: Setting only

Manual operation: See "Import File" on page 654

MMEMory:LOAD:EYE:BPATtern <TraceName>, <TraceFile>

Loads a user-defined bit pattern for the related eye diagram from a 7-bit ASCII file.

The pattern is repeated until the configured length is reached (see `CALCulate<Chn>:EYE:INPut:LENGth:BITS` on page 944).

Loading a pattern from file implicitly sets the type of bit stream to USER (see `CALCulate<Chn>:EYE:INPut:BPATtern:TYPE`).

Setting parameters:

<TraceName> Name of the related eye diagram

<TraceFile> String parameter containing the path and file name of the bit pattern file. If the path is omitted, the current directory is used (see `MMEMory:CDIRectory`).

Usage: Setting only

Options: R&S ZNB-K20

Manual operation: See "Load Bit Stream" on page 385

MMEMory:LOAD:EYE:EQUalization:CTLE <TraceName>, <TraceFile>

Loads a CTLE equalizer specification from a `.json` file.

Setting parameters:

<TraceName> Name of the related eye diagram.

<TraceFile> String parameter containing the path and name of the file with the CTLE configuration. If the path is omitted, the current directory is used.

Usage: Setting only

Options: R&S ZNB-K20

Manual operation: See "Load, Save" on page 395

MMEMory:LOAD:EYE:EQUalization:DFE <TraceName>, <TraceFile>

Loads a DFE equalizer specification from a `.json` file.

Setting parameters:

<TraceName> Name of the related eye diagram.

<TraceFile> String parameter containing the path and name of the file with the DFE configuration. If the path is omitted, the current directory is used.

Usage: Setting only

Options: R&S ZNB-K20

Manual operation: See "Load, Save" on page 397

MMEMory:LOAD:EYE:EQUalization:FFE <TraceName>, <TraceFile>

Loads a FFE equalizer specification from a `.json` file.

Setting parameters:

<TraceName> Name of the related eye diagram.

<TraceFile> String parameter containing the path and name of the file with the FFE configuration. If the path is omitted, the current directory is used.

Usage: Setting only

Options: R&S ZNB-K20

Manual operation: See "Load, Save" on page 396

MMEMory:LOAD:EYE:JITTer <TraceName>, <TraceFile>

Loads user-defined jitter from a 7-bit ASCII file into the generator simulation of an eye diagram.

The file must consist of floating point values (in parsable format), separated by any whitespace and/or line endings.

Setting parameters:

<TraceName> Name of the related eye diagram

<TraceFile> String parameter containing the path and file name of the jitter file. If the path is omitted, the current directory is used (see `MMEMory:CDIRectory`).

Usage: Setting only

Options: R&S ZNB-K20

Manual operation: See "User Specific" on page 392

MMEMory:LOAD:EYE:MASK <TraceName>, <TraceFile>

Loads a user-defined eye mask from a 7bit ASCII file.

Setting parameters:

<TraceName> Name of the related eye diagram

<TraceFile> String parameter containing the path and file name of the eye mask file. If the path is omitted, the current directory is used (see `MMEMory:CDIRectory`).

Usage: Setting only

Options: R&S ZNB-K20

Manual operation: See "Save / Load Mask Configuration" on page 401

MMEMory:LOAD:LIMit <TraceName>, <LimLineFile>[, <TouchstoneFile>[, <StimulusOffset>[, <ResponseOffset>[, <LimLineType>]]]]

Loads a limit line definition from a specified file and assigns it to a trace with a specified name. Limit lines are created using the `CALCulate<Ch>:LIMit...` commands.

Note: Limit lines can be loaded from Touchstone files (*.s<n>p, where <n> denotes the number of ports). The optional parameters `'<TouchstoneFile>'`, `<StimulusOffset>`, `<ResponseOffset>`, `<LimLineType>` are only relevant for Touchstone files. For *.limit files, no optional parameters can be set.

Setting parameters:

<TraceName> Name of an existing trace in the active recall set (string parameter). The imported limit line is assigned to this trace, irrespective of the trace information in the limit line file.

<LimLineFile> String parameter to specify the name and directory of the limit line file to be loaded. The default extension (manual control) for limit line files is *.limit, although other extensions are allowed. If no path is specified the analyzer searches the current directory, to be queried with `MMEMory:CDIRectory?`. See also note on Touchstone files above.

<TouchstoneFile> String parameter, selects an S-parameter from a Touchstone file. The parameter must be compatible with the file type (e.g. for one-port Touchstone files *.s1p, only the parameter name 'S11' is allowed).

*RST: 'S11' (if all optional parameters are omitted)

<StimulusOffset> Stimulus offset for limit lines loaded from a Touchstone file. A 1 GHz offset shifts the limit line by 1 GHz in (positive) horizontal direction.

Range: Depending on the sweep range of the analyzer.
*RST: 0
Default unit: NN

<ResponseOffset> Response offset for limit lines loaded from a Touchstone file. A 1 dB offset shifts the limit line by 1 dB in (positive) vertical direction.

Range: Depending on the measured quantity.
*RST: 0
Default unit: NN

<LimLineType> LMIN | LMAX | OFF

Limit line type :
LMAX - upper limit line
LMIN - lower limit line
OFF - limit line off

*RST: LMAX (if all optional parameters are omitted)

Example: Assume that the current recall set contains two traces named Trc1 and Trc2, respectively, and that limit lines have been defined for Trc1.

```
MMEM:STOR:LIM 'TRC1', 'C:
\Users\Public\Documents
\Rohde-Schwarz\Vna\LIMitLines\Lim_Trc1.limit'
```

Store the limit line definition of Trc1 to a limit line file.

```
MMEM:LOAD:LIM 'TRC2', 'C:
\Users\Public\Documents
\Rohde-Schwarz\Vna\LIMitLines\Lim_Trc1.limit'
```

Load the previously created limit line file and assign the limit lines to Trc2.

```
MMEM:STOR:TRAC 'TRC1', 'C:
\Users\Public\Documents
\Rohde-Schwarz\Vna\LIMitLines\Trc1.s1p'
```

Store the current trace data of Trc1 to a limit line file in Touchstone format.

```
MMEM:LOAD:LIM 'TRC1', 'C:
\Users\Public\Documents
\Rohde-Schwarz\Vna\LIMitLines\Trc1.s1p', 'S11',
0, 2, LMAX
```

Load the previously created Touchstone limit line file and assign the limit lines to Trc1, applying a response offset of 2 dB.

```
CALC:LIMit:DISPlay ON
```

Show the limit line in the diagram.

Usage: Setting only

Manual operation: See "Recall... / Save..." on page 490

MMEMory:LOAD:RIPPle <TraceName>, <RippleLimFile>

Loads a ripple limit definition from a specified file and assigns it to a trace with a specified name. Ripple limits are created using the `CALCulate<Ch>:RIPPle...` commands.

Setting parameters:

<TraceName> Name of an existing trace in the active setup (string parameter). The imported ripple limit line is assigned to this trace, irrespective of the trace information in the ripple limit file.

*RST: -

<RippleLimFile> String parameter to specify the name and directory of the ripple limit file to be loaded. The default extension (manual control) for ripple limit files is *.ripple, although other extensions are allowed. If no path is specified the analyzer searches the current directory, to be queried with `MMEMory:CDIRectory`

Example: Assume that the current setup contains two traces named Trc1 and Trc2, respectively, and that ripple limits have been defined for Trc1.

```
MMEM:STOR:RIPP 'TRC1', 'C:
\Users\Public\Documents
\Rohde-Schwarz\Vna\LIMitLines\Lim_Trc1.limit'
```

Store the ripple limit definition of Trc1 to a ripple limit file.

```
MMEM:LOAD:RIPP 'TRC2', 'C:
\Users\Public\Documents
Rohde-Schwarz\Vna\LIMitLines\Lim_Trc1.limit'
```

Load the previously created ripple limit file and assign the limits to Trc2.

```
CALC:RIPP:DISPlay ON
```

Show the ripple limit line for the active trace in the diagram.

Usage: Setting only

Manual operation: See "Recall Ripple Test.../Save Ripple Test..." on page 497

MMEMory:LOAD:SEGMent <Channel>, <SweepSegFile>

Replaces the related channel's current sweep segment definition by a sweep segment definition loaded from the specified ASCII file.

Setting parameters:

<Channel> Channel number

<SweepSegFile> String parameter to specify the name and directory of the sweep segment file to be loaded. The default extension (manual control) for sweep segment files is `*.SegList`, although other extensions are allowed.
If no path is specified the analyzer searches the current directory, to be queried with `MMEMory:CDIRectory`

Example: Assume that the current recall set contains two channels numbered 1 and 2, respectively, and that sweep segments have been defined for channel no. 1.
```
MMEM:STOR:SEGM 1, 'C:\Users\Public\Documents\
Rohde-Schwarz\Vna\SweepSegments\Seg_Ch1.SegList'
```
Store the sweep segment definition of channel 1 to a sweep segment file.
```
MMEM:LOAD:SEGM 2, 'C:\Users\Public\Documents\
Rohde-Schwarz\Vna\SweepSegments\Seg_Ch1.SegList'
```
Load the previously created sweep segment file and use the sweep segments for channel 2.

Usage: Setting only

Manual operation: See "Import/ Export" on page 548

MMEMory:LOAD:STATe <Compatibility>, <RecallSetFile>

Loads configuration data from a specified recall set file (`*.znx|*.znxml`) and sets the analyzer to the corresponding instrument state.

Setting parameters:

<Compatibility> 1 (this value is used for compatibility with the SCPI standard but is ignored).

<RecallSetFile> String parameter to specify the absolute or relative path of the recall set file to be loaded. Relative paths are evaluated relative to the current directory (see `MMEMory:CDIRectory`).

Example:
```
MMEM:STOR:STAT 1,'C:\Users\Public\Documents\
Rohde-Schwarz\Vna\RecallSets\Set_0413.znx'
```
Store the current setup configuration in the file `Set_0413.znx` in the default directory for recall set files.
```
MMEM:LOAD:STAT 1,'C:\Users\Public\Documents\
Rohde-Schwarz\Vna\RecallSets\Set_0413.znx'
```
Load the settings stored in `Set_0413.znx`.

Usage: Setting only

Manual operation: See "Open Recall..." on page 780

MMEMory:LOAD:TRACe <DestinationTraceName>, <TraceFile>[, <SParamOrTraceName>]

Loads trace data from a specified trace file and assigns it to a trace with a specified name. Traces are created using the `CALCulate<Ch>:PARameter:SDEFine` command.

Setting parameters:

<Destination TraceName> Name of an existing data trace in the active recall set (string parameter). The trace data is loaded into a memory trace associated with the specified data trace. If one or more memory traces are already associated with the specified data trace, the last generated memory trace is overwritten.

<TraceFile> String parameter to specify the name and directory of the trace file to be loaded. Several file formats for trace files are supported. The file extensions `*.s<n>p`, `*.csv`, and `*.dat` for Touchstone, ASCII, and Matlab files are mandatory.
If no path is specified the analyzer searches the current directory, to be queried with `MMEMory:CDIRectory`

<SParam OrTraceName> Optional string parameter: For imported Touchstone files for more than one port (`*.s2p`, `*.s3p`, `*.s4p`), the parameter denotes the imported S-parameter ('S11', 'S12', ...). For ASCII (`*.csv`) and Matlab (`*dat`) files, the parameter references a trace name in the file (case sensitive). If the parameter is omitted, the first trace in the specified file is imported.

Example: Assume that the current recall set contains a trace named Trc1.

```
MMEM:STOR:TRAC 'TRC1', 'C:\Users\Public
\Documents\Rohde-Schwarz\Vna\Traces\Trc1.s1p'
```

Store the current trace data of Trc1 to a trace file.

```
MMEM:LOAD:TRAC 'TRC1', 'C:\Users\Public
\Documents\Rohde-Schwarz\Vna\Traces\Trc1.s1p'
```

Load the previously created trace file and create a memory trace assigned to Trc1.

```
CALC:PAR:DEF:SGR 1,2
```

Create four traces to measure the two-port S-parameters S_{11}, S_{12}, S_{21}, S_{22}. The traces are not displayed.

```
MMEM:STOR:TRAC 'TRC1', 'C:\Users\Public
\Documents\Rohde-Schwarz\Vna\Traces\Trc1.s2p'
```

Store the four S-parameter traces to a two-port Touchstone file.

```
MMEM:LOAD:TRAC 'TRC1', 'C:\Users\Public
\Documents\Rohde-Schwarz\Vna\Traces\Trc1.s2p'
```

Load the previously created Touchstone file and overwrite the previously generated memory trace assigned to Trc1 with the S_{11} trace.

Usage: Setting only

Manual operation: See "Import Data to New Mem" on page 477

MMEMory:LOAD:TRACe:AUTO <TraceFile>

Loads the specified trace file and automatically distributes the imported S-parameter traces S_{ij} to all diagrams in the active channel that are currently displaying S_{ij}.

Setting parameters:

<TraceFile> String parameter to specify the name and directory of the trace file to be loaded. Several file formats for trace files are supported. The file extensions `*.s<n>p`, `*.csv`, and `*.dat` for Touchstone, ASCII, and Matlab files are mandatory.
If no path is specified the analyzer searches the current directory, to be queried with `MMEMory:CDIRectory`

Usage: Setting only

Manual operation: See "Auto Distribute" on page 482

MMEMory:LOAD:VNETworks<Ch>:BALanced:EMBedding<LogPt>
<TouchstoneFile>[, <Port>[, <Interchange>]]
MMEMory:LOAD:VNETworks<Ch>:BALanced:DEEMbedding<LogPt>
<TouchstoneFile>[, <Port>[, <Interchange>]]

Loads the S-parameter data that (partly) specify the de-/embedding network of balanced port `<LogPt>` from local Touchstone files.

Use `CALCulate<Ch>:TRANsform:VNETworks:BALanced:DEEMbedding<LogPt>:TNDefinition` or `CALCulate<Ch>:TRANsform:VNETworks:BALanced:EMBedding<LogPt>:TNDefinition`to select the desired circuit model **before** loading the data files.

- The `FIMPort` model requires a single `*.s4p` file and hence a single set command.
- The circuit models `STSL | STSC | SLST | SCST` require 2 `*.s2p` files, that have to be assigned to the appropriate port using two set commands.

Suffix:

<Ch> Channel number

<LogPt> Logical port number

Parameters:

<TouchstoneFile> String parameter to specify the name and directory of the loaded Touchstone file.
If no path is specified the analyzer searches the current directory, which can be set and queried using `MMEMory:CDIRectory`

<Port> PMAin | PSECondary

Assigns a 2-port (`*.s2p`) file to the appropriate port. PMAin corresponds to D1, PSECondary to D2 in the circuit model pictograms
For an `s4p` file, this parameter is ignored. However, it must be specified unless the <Interchange> parameter is also omitted.

<Interchange> FPORts | IPORts | SGATes | SINCreasing

Defines how to interpret the (implicit) port numbering of the Touchstone file in the context of the deembedding network.

FPORts (or omitted)
Standard interpretation: odd port numbers towards VNA, even port numbers towards DUT

IPORts
For `s2p` files IPORts means "inverted port sequence": network port 2 towards VNA, network port 1 towards DUT
For `s4p` files IPORts means "increasing port sequence": low port numbers towards VNA, high port numbers towards DUT

SGATes
Swapped gates: even port numbers towards VNA, odd port numbers towards DUT

SINCreasing
Swapped increasing port sequence: high port numbers towards VNA, low port numbers towards DUT

<SmoothingPercentage> Smoothing percentage (**deembedding only**), see "Smoothing/ Smoothing Percentage" on page 759.
If omitted, smoothing is disabled.

Range: 1 to 100
Increment: 1

Example:
```
*RST; SOUR:LPOR1 1,2; LPOR2 3,4
```
Define a balanced port configuration.
```
CALC:TRAN:VNET:BAL:DEEM:TND STSL
```
Select the "Serial .s2p data, shunt L" circuit model.
```
MMEM:LOAD:VNET:BAL:DEEM2 'C:\Users\Public
\Documents\Rohde-Schwarz\Vna\VNET\Test.s2p',
PMA
```
Load a Touchstone file and assign it to logical port no. 2.

Manual operation: See "File Name <i>/Inc. Seq. <i>/Swap Gates <i>" on page 711

MMEMory:LOAD:VNETworks<Ch>:DIFFerential:EMBedding<LogPt>
<TouchstoneFile>[, <Interchange>]

Loads data of a Differential match embedding network from the specified Touchstone `*.s2p` file.

Use `CALCulate<Ch>:TRANsform:VNETworks:BALanced:EMBedding<LogPt>:PARameters:DATA<Port>` to load circuit data from the remote client instead.

The query returns the name of the loaded file.

Suffix:
<Ch> Channel number

<LogPt> Logical port number of a balanced port

Parameters:

<TouchstoneFile> String parameter to specify the name and directory of the loaded `*.s2p` file.
If no path is specified the analyzer searches the current directory, which can be set and queried using `MMEMory:CDIRectory`

<Interchange> FPORts | IPORts | SGATes

FPORts (or omitted)
Standard port sequence (network port 1 towards VNA, network port 2 towards DUT)

IPORts | SGATes
Inverted port sequence (network port 2 towards VNA, network port 1 towards DUT)

Example:
```
MMEM:LOAD:VNET1:DIFF:EMBM1 'C:\Users\Public\Documents\Rohde-Schwarz\Vna\Embedding\Test.s2p'
```
Load a Touchstone file.

Manual operation: See "File Name 1/Swap Gates" on page 714

MMEMory:LOAD:VNETworks<Ch>:GLOop:DEEMbedding<group> <TouchstoneFile>
MMEMory:LOAD:VNETworks<Ch>:GLOop:EMBedding<group> <TouchstoneFile>

Loads data from a specified one-port (`*.s1p`) Touchstone file defining a ground loop circuit model for de-/embedding.

Suffix:

<Ch> Channel number

<group> Port group (DUT) number.
If multiple port groups are configured (see `SOURce<Ch>:GROup<Grp>:PPORts`) and `CALCulate<Ch>:TRANsform:VNETworks:GLOop:GROup` is `ON`, then each port group can have its own ground loop de-/embedding model.

Parameters:

<TouchstoneFile> String parameter to specify the name and directory of the loaded Touchstone file.
If no path is specified the analyzer searches the current directory, which can be set and queried using `MMEMory:CDIRectory`

Example:
```
CALC:TRAN:VNET:SEND:GLO:TND FIMP
```
Select the 1-Port Data (s1p) circuit model.
```
MMEM:LOAD:VNET:GLO:EMB 'C:\Users\Public\Documents\Rohde-Schwarz\Vna\Embedding\Test.s1p'
```
Load a Touchstone file.

Manual operation: See "File Name 1" on page 712

MMEMory:LOAD:VNETworks<Ch>:PPAir:EMBedding<ListId> <TouchstoneFile>[, <Port>[, <Interchange>]]
MMEMory:LOAD:VNETworks<Ch>:PPAir:DEEMbedding<ListId> <TouchstoneFile>[, <Port>[, <Interchange>]]

Loads the S-parameter data that (partly) specify the deembedding network of port group `<ListId>` from local Touchstone files.

Use `CALCulate<Ch>:TRANsform:VNETworks:PPAir:DEEMbedding<ListId>:TNDefinition` or `CALCulate<Ch>:TRANsform:VNETworks:PPAir:EMBedding<ListId>:TNDefinition` to select the desired circuit model **before** loading the data files.

- For any n-port set, the `FIMPort` model requires a single `*.s<2n>p` file and hence a single set command.
- For 2-port sets (port pairs), the circuit models `STSL | STSC | SLST | SCST` require 2 `*.s2p` files, that have to be assigned to the appropriate port using two set commands.

Suffix:

<Ch> Channel number

<ListId> Index of the port set within the channel's overall list of port sets for de-/embedding.

Parameters:

<TouchstoneFile> String parameter specifying the name and directory of the loaded Touchstone file.
If no path is specified, the analyzer searches the current directory, which can be set and queried using `MMEMory:CDIRectory`

<Port> PMAin | PSECondary

For port pairs, <Port> assigns a 2-port (`*.s2p`) file to the appropriate port. PMAin corresponds to D1, PSECondary to D2 in the circuit model pictograms for balanced ports
For an `s<2m>p` file with m>1, this parameter is ignored. However, it must be specified unless the <Interchange> parameter is also omitted.

<Interchange> FPORts | IPORts | SGATes | SINCreasing

Defines how to interpret the (implicit) port numbering of the Touchstone file in the context of the de-/embedding network.

FPORts (or omitted)
Standard interpretation: odd port numbers towards VNA, even port numbers towards DUT

IPORts
For `s2p` files IPORts means "inverted port sequence": network port 2 towards VNA, network port 1 towards DUT
For `s4p` files IPORts means "increasing port sequence": low port numbers towards VNA, high port numbers towards DUT

SGATes
Swapped gates: even port numbers towards VNA, odd port numbers towards DUT

SINCreasing
Swapped increasing port sequence: high port numbers towards VNA, low port numbers towards DUT

<SmoothingPercentage> Smoothing percentage (**deembedding only**), see "Smoothing/ Smoothing Percentage" on page 759.
If omitted, smoothing is disabled.

Range: 1 to 100
Increment: 1

Example: `*RST; :CALC:TRAN:VNET:PPA:DEEM:DEF 1,2,3,4`
Define a port pair configuration with port pairs (1,2) and (3,4).
`CALC:TRAN:VNET:PPA:DEEM2:TND STSL`
Select the *Serial Touchstone .s2p data, shunt L* circuit mode for the second port pair.
`MMEM:LOAD:VNET:PPA:DEEM2 'C:\Rohde-Schwarz\Vna\Traces\Test.s2p', PMA`
Load a Touchstone file and assign it to the second port pair.

Manual operation: See "File Name <i>/Inc. Seq. <i>/Swap Gates <i>" on page 710

MMEMory:LOAD:VNETworks<Ch>:SENDed:EMBedding<PhyPt>
<TouchstoneFile>[, <Interchange>]
MMEMory:LOAD:VNETworks<Ch>:SENDed:DEEMbedding<PhyPt>
<TouchstoneFile>[, <Interchange>[, <SmoothingPercent>]]

Loads data from a specified two-port (`*.s2p`) Touchstone file defining a single ended circuit model for de-/embedding.

Use

- `CALCulate<Ch>:TRANsform:VNETworks:SENDed:DEEMbedding<PhyPt>:TNDefinition` or `CALCulate<Ch>:TRANsform:VNETworks:SENDed:EMBedding<PhyPt>:TNDefinition` to select the adequate circuit model **before** executing this command.
- `CALCulate<Ch>:TRANsform:VNETworks:SENDed:EMBedding<PhyPt>:PARameters:DATA` to load circuit data from the remote client.

Suffix:
<Ch> Channel number

<PhyPt> Physical port number

Parameters:

<TouchstoneFile> String parameter to specify the name and directory of the loaded Touchstone file.
If no path is specified, the analyzer searches the current directory, which can be set and queried using `MMEMory:CDIRectory`.

<Interchange> FPORts | IPORts | SGATes

FPORts (or omitted)
Standard port sequence (network port 1 towards VNA, network port 2 towards DUT)

IPORts | SGATes
Inverted port sequence (network port 2 towards VNA, network port 1 towards DUT)

<SmoothingPercent> Smoothing percentage (**deembedding only**), see "Smoothing/Smoothing Percentage" on page 759.
If omitted, smoothing is disabled.

Range: 1 to 100
Increment: 1

Example:

```
CALC:TRAN:VNET:SEND:DEEM:TND FIMP
```
Select the "Serial .s2p" data circuit model.
```
MMEM:LOAD:VNET:SEND:DEEM2 'C:\Users\Public\Documents\Rohde-Schwarz\Vna\VNET\Test.s2p'
```
Load a Touchstone file and assign it to the physical port no. 2.

Manual operation: See "File Name 1/Swap Gates" on page 708

MMEMory:MDIRectory <NewDirectory>

Creates a new subdirectory for mass memory storage in an existing directory.

Setting parameters:

<NewDirectory> String parameter to specify the new directory. Either the full path or a subdirectory for the current directory (see `MMEMory:CDIRectory`).

Example:

```
MMEM:MDIR 'C:\Users\Public\New_Directory'
```
Create the specified directory. The parent directory `C:\Users\Public` must have been created before.
```
MMEM:MDIR 'C:\Users\Public\New_Directory\New_Subdirectory'
```
Create an additional subdirectory.
```
MMEM:CDIR 'C:\Users\Public\Instrument'; MDIR 'New_Directory'
```
Create an additional directory `C:\Users\Public\Instrument\New_Directory`.

Usage: Setting only

MMEMory:MOVE <SourceFile>, <NewFile>

Moves a file to the indicated directory and stores it under the file name specified, if any. If `<NewFile>` contains no path indication, the command renames the file without moving it.

Setting parameters:
<SourceFile>

<NewFile> String parameters to specify the name and the path of the file to be copied and the name and the path of the new file.

Example:
```
MMEM:MOVE 'C:\Users\Public\Documents
\Rohde-Schwarz\Vna\RecallSets\SET1.znx','D:'
```
Move file `Set1.znx` in directory `C:\Users\Public\Documents\Rohde-Schwarz\Vna\RecallSets` to an external storage medium, mapped to drive `D:`.

Usage: Setting only

MMEMory:MSIS <Drive>

Sets/gets the *current drive* for `MMEMory` commands (MSIS = mass storage identification string).

Other `MMEMory` commands interpret paths starting with a "\" relative to this drive.

`MMEMory:MSIS <Drive>` is equivalent to `MMEMory:CDIRectory <Drive>`. In particular, it sets the current directory to the base directory of the specified drive.

Parameters:
<Drive> Drive letter, followed by a colon, e.g. `'D:'`

Example:
```
MMEMory:CDIRectory DEFault
```
selects the default directory `C:\Users\Public\Documents\Rohde-Schwarz\Vna`.
```
MMEMory:MSIS?
```
returns `C:`.
```
MMEMory:CATalog? 'hardcopy'
```
lists the contents of `C:\Users\Public\Documents\Rohde-Schwarz\Vna\hardcopy`.
```
MMEMory:CATalog? '\hardcopy'
```
lists the contents of `D:\hardcopy`.

MMEMory:NAME <Filename>

Defines a name for a file which can be used to store the printer output. The file is created when it is selected as a printer destination (`HCOPy:DESTination 'MMEM'`).

Parameters:

<Filename> String parameter to specify the file name. The supported file formats are `*.wmf`, `*.ewmf`, `*.bmp`, `*.png`; see command `HCOPy:DEVice:LANGuage`. The specified directory must exist, otherwise no file can be generated. If no path is specified the analyzer uses the current directory, to be queried with `MMEMory:CDIRectory?`.

*RST: 'Hardcopy'

Example:

```
MMEM:NAME 'C:
\Users\Public\Screenshots\PLOT1.BMP'
```

Define a printer file name, to be stored in the existing directory `C:\Users\Public\Screenshots` (without creating the file).

```
HCOP:DEST 'MMEM'; :HCOP
```

Select "Print to file" and create the printer file specified before.

MMEMory:RDIRectory <Directory>

Allows you to remove an **empty** directory from the mass memory storage system.

Setting parameters:

<Directory> String parameter to specify the directory.

Example:

```
MMEM:RDIR 'C:
\Users\Public\NetworkService\Application Data'
```

Removes the specified directory, if empty. Otherwise an error occurs.

Usage: Setting only

MMEMory:SIMulation:DELete:ALL

R&S ZNXSIM only.

Deletes all import rules for additional simulation data, specified using `MMEMory:SIMulation:SDATa<Idx>:LOAD`.

Usage: Event

Manual operation: See "Delete/Delete All" on page 791

MMEMory:SIMulation:LOAD <FileName>, <Channel>

R&S ZNXSIM only.

Loads simulation data from file.

Setting parameters:

<FileName> Path to a trace file (`*.sNp, *.csv, *.dat`), either absolute or relative to the current directory (`MMEMory:CDIRectory`)

<Channel> You can either load simulation data into a particular channel (identified by its channel number) or into ALL channels.

Usage: Setting only

Manual operation: See "Load Simulation Data..." on page 787

MMEMory:SIMulation:SDATa<Idx> <PortMapping>, <Parameters>

R&S ZNXSIM only.

Modifies an import rule for additional simulation data, specified using `MMEMory:SIMulation:SDATa<Idx>:LOAD`.

Suffix:
<Idx> The index/position ot the rule to be modified.

Setting parameters:
<PortMapping> Modified port mapping
String parameter, same syntax as in `MMEMory:SIMulation:SDATa<Idx>:LOAD`.

<Parameters> Modified selection of S-parameters
String parameter, same syntax as in `MMEMory:SIMulation:SDATa<Idx>:LOAD`.

Usage: Setting only

Manual operation: See "Step 3: Select S-parameters" on page 791

MMEMory:SIMulation:SDATa<Idx>:DELete

R&S ZNXSIM only.

Deletes an import rule for additional simulation data, specified using `MMEMory:SIMulation:SDATa<Idx>:LOAD`.

Suffix:
<Idx> The index/position ot the rule to be deleted.

Usage: Event

Manual operation: See "Delete/Delete All" on page 791

MMEMory:SIMulation:SDATa<Idx>:LOAD <TouchstoneFile>, <PortMapping>, <Parameters>

R&S ZNXSIM only.

Creates a rule for importing S-parameter traces from a Touchstone file into the active channel's simulation data. Then executes the rules in ascending <Idx> order.

Suffix:
<Idx> Insert position of the import rule.
If no index is specified, the rule is added at the end.

Setting parameters:

<TouchstoneFile> String parameter to specify the name and directory of the Touchstone file to be loaded.
If no path is specified, the analyzer searches the current directory (`MMEMory:CDIRectory`)
.

<PortMapping> String parameter mapping physical ports to data file ports
Specified in the format `'{phys. port1},{data port1},{phys. port2},{data port2}, ...'`. E.g., `'1,3,2,4'` indicates that physical port 1 is assigned to data file port 3 and physical port 2 is assigned to data file port 4.

<Parameters> The S-parameters to be overwritten (using phys. port numbers), E.g., `'S12,S21'` means that the VNA's S_{12} and S_{21} are overwritten by the corresponding S-parameters in the data file (S34 and S43 in the port mapping example above).

Usage: Setting only

Manual operation: See "Step 3: Select S-parameters" on page 791

MMEMory:STORe:CABLe <OutputDir>

Saves all predefined and user-defined Distance to Fault (DtF) cable types to the specified output directory.

Each DtF cable type is stored in a separate ASCII file `<cable name>.rsc`. The command silently overwrites files of the same name previously existing in the output directory.

Setting parameters:

<OutputDir> Output Directory, must be created before the command is executed

Example: see `MMEMory:LOAD:CABLe` on page 1261

Usage: Setting only

Options: R&S ZNB-K3

Manual operation: See "Add / Delete" on page 372

MMEMory:STORe:CKIT <CalKitName>, <CalKitFile>

Stores the data of a calibration kit to a specified file. The calibration kit is identified by its name.

Setting parameters:

<CalKitName> Name of a user-defined calibration kit available on the analyzer.
Tip: It is not possible to modify or store predefined or ideal kits.

<CalKitFile> String parameter to specify the name and directory of the cal kit file to be created. The file is a network analyzer-specific cal kit file with the extension `*.calkit`.
If no path is specified the analyzer searches the current directory, to be queried with `MMEMory:CDIRectory`

Example:
```
MMEM:LOAD:CKIT 'C:\Users\Public\Documents
\Rohde-Schwarz\Vna\Calibration\Kits\New Kit
1.calkit'
```
Load the previously created cal kit file `New Kit 1.calkit` from the default cal kit directory.
```
... :MMEM:STOR:CKIT 'New Kit 1', 'C:
\Users\Public\Documents
\Rohde-Schwarz\Vna\Calibration\Kits\New Kit
1.calkit'
```
Store the data for the user-defined cal kit "New Kit 1" and overwrite the cal kit file `New Kit 1.calkit`.

Usage: Setting only

Manual operation: See "Import Cal Kit / Export Cal Kit" on page 639

MMEMory:STORe:CKIT:WLABel <CalKitName>, <KitLabel>, <CalKitFile>

Stores the data of a calibration kit to a specified file. The calibration kit is identified by its name and label.

Setting parameters:

<CalKitName> Name of a user-defined calibration kit available on the analyzer.
Tip: It is not possible to modify or store predefined or ideal kits.

<KitLabel> Label of the calibration kit, usually its serial number.

<CalKitFile> String parameter to specify the name and directory of the cal kit file to be created. The file is a NWA-specific cal kit file with the extension *.calkit.
If no path is specified the analyzer searches the current directory, to be queried with `MMEMory:CDIRectory`

Example: See `[SENSe:]CORRection:CKIT:LLABel`

Usage: Setting only

MMEMory:STORe:CMAP <ColorSchemeFile>
MMEMory:STORe:CMAP:HCOPy <ColorSchemeFile>

Stores the user-defined color scheme (display/print) to a VNA color scheme file.

The `HCOPy` variant is for the print color scheme.

Setting parameters:

<ColorSchemeFile> String parameter to specify the name and directory of the color scheme file to be created. If no path is specified the analyzer uses the current directory, to be queried with `MMEMory:CDIRectory?`. The default extension (manual control) for color scheme files is `*.ColorScheme`, although other extensions are allowed.

Example: See `MMEMory:LOAD:CMAP`

Usage: Setting only

Manual operation: See "Recall... / Save..." on page 806

MMEMory:STORe:CORRection <Channel>, <CalGroupFile>

Copies the correction data of channel `<Channel>` to the cal pool, generating a new correction data file (cal group). The file has the extension `*.cal` and is stored in the `C:\Users\Public\Documents\Rohde-Schwarz\Vna\Calibration\Data` directory.

Setting parameters:

<Channel> Channel number

<CalGroupFile> String parameter to specify the name of the created cal group file. There is no need to specify the directory path and file extension; the analyzer uses the default cal pool directory `C:\Users\Public\Documents\Rohde-Schwarz\Vna\Calibration\Data` and a `*.cal` extension.

Example: See `MMEMory:LOAD:CORRection`

Usage: Setting only

Manual operation: See "Add / Add All / Replace / Apply / Apply to All" on page 658

MMEMory:STORe:CORRection:TCOefficient<Ch> <PmclFile>

Saves two-port transmission coefficients of channel <Ch> to a power meter correction list file.

Suffix:

<Ch> Channel number

Setting parameters:

<PmclFile> String parameter specifying the name and directory of the created power meter correction list file. The file extension `*.pmcl` is mandatory.
If no path is specified, the analyzer searches the current directory, to be queried with `MMEMory:CDIRectory`

*RST: n/a

<DeEmbeddingPosition> BOTH | DUT | PM

Determines the two-port the command refers to:

DUT
The two-port between VNA and DUT (during measurement)

PM
The two-port between VNA and PM (during power calibration)

BOTH
Both positions

(parameter omitted)
If the parameter is omitted, the command refers to the two-port selected using `SOURce<Ch>:POWer<PhyPt>:CORRection:TCOefficient:CALibration`.

Example: See `SOURce:POWer:CORRection:TCOefficient[:STATe]`

Usage: Setting only

Manual operation: See "Recall / Save" on page 655

MMEMory:STORe:EYE:EQUalization:CTLE <TraceName>, <TraceFile>

Stores the CTLE equalizer specification in a `.json` file.

Setting parameters:

<TraceName> Name of the related eye diagram.

<TraceFile> String parameter specifying the path and name of the file storing the CTLE configuration. If the path is omitted, the current directory is used.

Usage: Setting only

Options: R&S ZNB-K20

Manual operation: See "Manual, Automatic" on page 395

MMEMory:STORe:EYE:EQUalization:DFE <TraceName>, <TraceFile>

Stores the DFE equalizer specification in a `.json` file.

Setting parameters:

<TraceName> Name of the related eye diagram.

<TraceFile> String parameter specifying the path and name of the file storing the DFE configuration. If the path is omitted, the current directory is used.

Usage: Setting only

Options: R&S ZNB-K20

Manual operation: See "Load, Save" on page 397

MMEMory:STORe:EYE:EQUalization:FFE <TraceName>, <TraceFile>

Stores the FFE equalizer specification in a `.json` file.

Setting parameters:

<TraceName> Name of the related eye diagram.

<TraceFile> String parameter specifying the path and name of the file storing the FFE configuration. If the path is omitted, the current directory is used.

Usage: Setting only

Options: R&S ZNB-K20

Manual operation: See "Load, Save" on page 396

MMEMory:STORe:EYE:MASK <TraceName>, <TraceFile>

Stores a user-defined eye mask to a 7bit ASCII file.

Setting parameters:

<TraceName> Name of the related eye diagram

<TraceFile> String parameter containing the path and file name of the eye mask file. If the path is omitted, the current directory is used (see `MMEMory:CDIRectory`).

Usage: Setting only

Options: R&S ZNB-K20

Manual operation: See "Save / Load Mask Configuration" on page 401

MMEMory:STORe:EYE:MASK:RESults <TraceName>, <TraceFile>

Saves the detailed results of the mask test in the related eye diagram to an ASCII file (see `CALCulate<Chn>:EYE:MASK:DATA?` on page 952).

This command raises an execution error if `CALCulate<Chn>:EYE:MASK:STATe` is `OFF`.

Setting parameters:

<TraceName> The name of the eye diagram whose mask test results shall be exported

<TraceFile> Mandatory string parameter containing the path and file name of the file. If the path is omitted, the current directory is used (see `MMEMory:CDIRectory`).

Usage: Setting only

Options: R&S ZNB-K20

Manual operation: See "Export Test Results" on page 398

MMEMory:STORe:EYE:MEASurements <TraceName>, <TraceFile>[, <DecSeparator>[, <FieldSeparator>]]

Allows you to export eye diagram results to an ASCII file (csv), with the possibility to specify a decimal separator (comma or point) and a field separator (semicolon, comma, tab or space).

See also `CALCulate<Chn>:EYE:MEASurement:DATA?` on page 959.

Note that the decimal separator and field separator must be different: if both are set to comma, actually a semicolon will be used as field separator.

Setting parameters:

<TraceName> The name of the eye diagram whose results shall be exported.

<TraceFile> Mandatory string parameter containing the path and file name of the file. If the path is omitted, the current directory is used (see `MMEMory:CDIRectory`).

<DecSeparator> POINt | COMMa
Decimal separator
*RST: COMMa

<FieldSeparator> SEMicolon | COMMa | TABulator | SPACe
Field separator
*RST: SEMicolon

Usage: Setting only

Options: R&S ZNB-K20

Manual operation: See "Export Measurements..." on page 383

MMEMory:STORe:LIMit <TraceName>, <LimLineFile>

Saves the limit lines associated to a specified trace to a limit line file. Limit lines are created using the `CALCulate<Chn>:LIMit...` commands.

Setting parameters:

<TraceName> Name of an existing trace in the active recall set (string parameter) for which a limit line definition exists.

<LimLineFile> String parameter to specify the name and directory of the created limit line file. The default extension (manual control) for limit line files is *.limit, although other extensions are allowed.
If no path is specified the analyzer searches the current directory, to be queried with `MMEMory:CDIRectory`

Example: See `MMEMory:LOAD:LIMit`

Usage: Setting only

Manual operation: See "Recall... / Save..." on page 490

MMEMory:STORe:MARKer <AsciiFile>

Saves the values of all markers to an ASCII file.

Setting parameters:

<AsciiFile> String parameter to specify the name and directory of the created ASCII file. The default extension (manual control) for marker files is *.txt, although other extensions are allowed.
If no path is specified the analyzer searches the current directory, to be queried with `MMEMory:CDIRectory`

Example:

```
*RST
```
Reset the analyzer, creating the default trace no. 1 in channel no. 1.
```
CALC:MARK ON; MARK:X 1GHz
```
Create marker no. 1 and place it to 1 GHz.
```
CALC:MARK2 ON; MARK2:X 2GHz
```
Create a second marker and place it to 2 GHz.
```
MMEM:STOR:MARK 'Marker.txt'
```
Store the marker values to an ASCII file. The file contains both marker values, e.g.:
```
Trc1 S21
Mkr 1 1.000000 GHz -4.900 dB
Mkr 2 2.000000 GHz -6.807 dB
```

Usage: Setting only

Manual operation: See "Export Markers" on page 508

MMEMory:STORe:RIPPle <TraceName>, <RippleLimFile>

Saves the ripple limits associated with a specified trace to a ripple limit file. Ripple limit definitions are created using the `CALCulate<Chn>:RIPPle...` commands.

Setting parameters:

<TraceName> Name of an existing trace in the active setup (string parameter) for which a ripple limit definition exists.

<RippleLimFile> String parameter to specify the name and directory of the created ripple limit file. The default extension (manual control) for ripple limit files is `*.ripple`, although other extensions are allowed.
If no path is specified the analyzer searches the current directory, to be queried with `MMEMory:CDIRectory`

Example: See `MMEMory:LOAD:LIMit`

Usage: Setting only

Manual operation: See "Recall Ripple Test.../Save Ripple Test..." on page 497

MMEMory:STORe:SEGMent <Channel>, <SweepSegFile>

Saves the sweep segment definition of the related channel to a an ASCII file. Sweep segments are defined using [SENSe:]SEGMent... commands.

Setting parameters:

<Channel> Channel number

<SweepSegFile> String parameter to specify the name and directory of the created sweep segment file. The default extension (manual control) for sweep segment files is `*.SegList`, although other extensions are allowed.
If no path is specified the analyzer searches the current directory, to be queried with `MMEMory:CDIRectory`

Example: See `MMEMory:LOAD:SEGMent`

Usage: Setting only

Manual operation: See "Import/ Export" on page 548

MMEMory:STORe:STATe <Compatibility>, <RecallSetFile>

Stores the configuration data of the current recall set to the specified recall set file.

`MMEMory:STORe:STATe` also renames the current recall set. See example for `MMEMory:LOAD:STATe`.

Setting parameters:

<Compatibility> 1 (this value is used for compatibility with the SCPI standard but ignored).

<RecallSetFile> String parameter to specify the absolute or relative path of the created recall set file. Relative paths are evaluated relative to the current directory (see `MMEMory:CDIRectory`).
The default extension for recall set files is `znx`. Unless you specify `znxml` as file name extension, recall sets are always stored in `znx` file format.

Example: See `MMEMory:LOAD:STATe`

Usage: Setting only

Manual operation: See "Save" on page 781

MMEMory:STORe:TRACe <TraceName>, <TraceFile>[, <FormatInd>[, <Format>[, <DecSeparator>[, <FieldSeparator>[, <DcExtrapolate>]]]]]

Stores the trace data of a specified data trace to a trace file. Traces are created using the `CALCulate<Ch>:PARameter:SDEFine` command.

Tip: `*.s<n>p` Touchstone files (<n> = 1, 2, 3, ...) are intended for a complete set of <n>-port S-parameters. Data export fails if the active channel does not contain the full set of $<n>^2$ traces or if the involved ports are not numbered consecutively, starting with port 1. If the necessary traces are available, `'<trc_name>'` can be the name of any of the traces. To create Touchstone files while less than $<n>^2$ single-ended traces are available, use `MMEMory:STORe:TRACe:PORTs`.

Setting parameters:

<TraceName> Name of an existing data trace in the active recall set (string parameter).

<TraceFile> String parameter to specify the name and directory of the created trace file. Several file formats for trace files are supported. The file extensions `*.s<n>p`, `*.csv`, and `*.dat` for Touchstone, ASCII, and MATLAB files are mandatory. To generate a multiport Touchstone file `*.s2p`, `*.s3p`..., the channel must contain traces for the full set of S-parameters; the `'<trc_name>'` is ignored. If no path is specified the analyzer uses the current directory, to be queried with `MMEMory:CDIRectory?`.

<FormatInd> FORMatted | UNFormatted

FORMatted
Formatted data export (`*.csv` and `*.dat` files only).

UNFormatted
Unformatted data export

If omitted, UNFormatted is assumed.

<Format> COMPlex | LINPhase | LOGPhase

COMPlex
Complex values (real and imaginary part)

LINPhase
Linear magnitude and phase (`*.csv` and `*.dat` files only)

LOGPhase
dB magnitude and phase (`*.csv` and `*.dat` files only)

If omitted, COMPlex is assumed.

<DecSeparator> POINt | COMMa

POINt
Points (dots) are used as decimal separators.

COMMa
Commas are used as decimal separators (`*.csv` and `*.dat` files only)

If omitted, POINt is assumed.

<FieldSeparator> SEMicolon | COMMa | TABulator | SPACe

SEMicolon
Semicolons are used as field separators.

COMMa
Commas are used as field separators.

TABulator
Tabulators are used as field separators.

SPACe
Spaces are used as field separators.

If omitted, SEMicolon is assumed.

<DcExtrapolate> DCPoint | NODCpoint

DCPoint
DC point extrapolation enabled

NODCpoint
DC point extrapolation disabled

If omitted, NODCpoint is assumed.

Example: See `MMEMory:LOAD:TRACe`

Usage: Setting only

Manual operation: See "Save" on page 481

MMEMory:STORe:TRACe:CHANnel <Channel>, <TraceFile>[, <FormatInd>[, <Format>[, <DecSeparator>[, <FieldSeparator>]]]]

Stores the trace data of all data traces in the specified channel to a trace file. Traces are created using the `CALCulate<Ch>:PARameter:SDEFine` command.

Tip: `*.s<n>p` Touchstone files (<n> = 1, 2, 3, ...) are intended for a complete set of <n>-port S-parameters. Data export fails if the active channel does not contain the full set of $<n>^2$ traces.

Setting parameters:

<Channel> Channel number in the active recall set. ALL means that a separate file is created for each channel in the active recall set.

<TraceFile> String parameter to specify the name and directory of the created trace file. Several file formats for trace files are supported. The file extensions `*.s<n>p`, `*.csvc`, and `*.dat` for Touchstone, ASCII, and Matlab files are mandatory. To generate a multiport Touchstone file `*.s2p`, `*.s3p...`, the channel must contain traces for the full set of S-parameters. If no path is specified the analyzer uses the `C:\Users\Public\Documents\Rohde-Schwarz\Vna\Traces` directory.

<FormatInd> FORMatted | UNFormatted

UNFormatted - unformatted data export specified by the second optional parameter.
FORMatted - formatted data export (for `*.csv` and `*.dat` files only).
If the first optional parameter is omitted, the command stores unformatted data.

<Format> COMPlex | LINPhase | LOGPhase

COMPlex - complex values (real and imaginary part)
LINPhase - linear magnitude and phase.
LOGPhase - dB-magnitude and phase.
If the second optional parameter is omitted, the command stores complex data.

<DecSeparator> POINt | COMMa

POINt - Decimal separator: point.
COMMa - Decimal separator: comma.
If the third optional parameter is omitted, points are used.

<FieldSeparator> SEMicolon | COMMa | TABulator | SPACe

SEMicolon - Field separator: semicolon
COMMa - field separator: comma.
TABulator - field separator: tabulator.
SPACe - field separator: space.
If the third optional parameter is omitted, semicolons are used.

Example:

```
*RST; :CONF:TRAC:NAME?
```

Reset the instrument, creating a default channel no 1 and a default trace Trc1.

```
CALC:PAR:DEF:SGR 1,2
```

Create four traces to measure the two-port S-parameters S_{11}, S_{12}, S_{21}, S_{22}. The traces are not displayed.

```
MMEM:STOR:TRAC:CHAN 1, 'C:\Users\Public
\Documents\Rohde-Schwarz\Vna\Traces\Chn1.csv'
```

Store all trace data of channel 1 to a trace file.

```
MMEM:STOR:TRAC:CHAN 1, 'C:\Users\Public
\Documents\Rohde-Schwarz\Vna\Traces\Chn1.s2p'
```

Store the four S-parameter traces to a two-port Touchstone file. The Touchstone file will not contain the default trace Trc1.

Usage: Setting only

Manual operation: See "Save" on page 481

MMEMory:STORe:TRACe:OPTion:BALanced <Boolean>
MMEMory:STORe:TRACe:OPTion:SYMMetric <Boolean>

This command enables/disables the export of balanced (and mixed-mode) S-parameters for `MMEMory:STORe:TRACe:PORTs`.

Parameters:
<Boolean>

MMEMory:STORe:TRACe:OPTion:COMMent <Comment>

Defines a comment to be added to (the comment section of) exported trace files.

This is a global setting, i.e. the comment string is used for each trace export in every open recall set.

Parameters:
<Comment> Comment string. If empty, no comment will be added.

Manual operation: See "Comment Added to File" on page 482

MMEMory:STORe:TRACe:OPTion:DECimals:DATA <NumDecimals>
MMEMory:STORe:TRACe:OPTion:DECimals:STIMulus <NumDecimals>

Defines the number of decimal places for stimulus and data values in all supported trace data files (Touchstone, ASCII, MatLab)

Note that in the current implementation of the Touchstone file export, only the `L11` file format supports independent settings for stimulus and data values (see `MMEMory:STORe:TRACe:OPTion:FORMat`). For the `F11` and `F20` formats, the `DATA` number of digits is used for both stimulus and data values.

Parameters:
<NumDecimals> Range: 1 to 15

Manual operation: See "Decimal Places" on page 480

MMEMory:STORe:TRACe:OPTion:EXTRapolate <Boolean>

This command enables/disables DC point extrapolation for `MMEMory:STORe:TRACe:PORTs`.

Parameters:
<Boolean>

ON (1)
DC point extrapolation enabled

OFF (0)
DC point extrapolation disabled

*RST: OFF

Manual operation: See "DC Point Extrapolation" on page 479

MMEMory:STORe:TRACe:OPTion:FORMat <TS_Frmt>

Selects the file format to be generated during Touchstone export.

Parameters:
<TS_Frmt> F11 | F20 | L11

F11
Trace output according to Touchstone standard 1.1 (new library)

F20
Trace output according to Touchstone standard 2.0 (new library)

L11
Legacy Touchstone 1.1 output

F11 and L11 slightly differ w.r.t. casing, number formatting and comments. F20 adds keywords etc. introduced with Touchstone standard version 2.0.

To optimize speed, use F20.

Manual operation: See "File Format" on page 479

MMEMory:STORe:TRACe:OPTion:PLUS <arg0>

This command defines how positive numbers are prefixed during Touchstone file export: by a leading space, a plus sign or not at all.

Parameters:

<arg0> SPACe | PLUS | VOID

Manual operation: See "Positive Number Prefix" on page 819

MMEMory:STORe:TRACe:OPTion:RSAMple <State>

Enables resampling of time domain traces during export. The sampling rate can be specified using `MMEMory:STORe:TRACe:OPTion:RSAMple:POINts` or `MMEMory:STORe:TRACe:OPTion:RSAMple:TIMStep`.

Resampling only applies if single time domain trace is exported in ASCII format (`*.csv`) and with `FORMatted` values (see `MMEMory:STORe:TRACe`).

Parameters:

<State>

OFF (0)
Resampling disabled

ON (1)
Resampling enabled

*RST: OFF

Options: R&S ZNB-K2

Manual operation: See "Resample/Resample Settings" on page 481

MMEMory:STORe:TRACe:OPTion:RSAMple:POINts <Points>

Defines the number of points for resampling time domain traces during export. See `MMEMory:STORe:TRACe:OPTion:RSAMple`.

Alternatively, you can set the time step size (see `MMEMory:STORe:TRACe:OPTion:RSAMple:TIMStep`).

Parameters:

<Points>

Number of time domain points of the resampled trace.
The value must be between 1 and the number of points of the underlying frequency sweep (default; see `[SENSe<Ch>:]SWEep:POINts`).
Use `MMEMory:STORe:TRACe:OPTion:RSAMple:PRESet` to set the resampling parameters to their default values.

Options: R&S ZNB-K2

Manual operation: See "Resample/Resample Settings" on page 481

MMEMory:STORe:TRACe:OPTion:RSAMple:PRESet

Resets the resampling parameters for time domain trace export to their defaults.

See `MMEMory:STORe:TRACe:OPTion:RSAMple:POINts` and `MMEMory:STORe:TRACe:OPTion:RSAMple:TIMStep`.

Usage: Event

Options: R&S ZNB-K2

Manual operation: See "Resample/Resample Settings" on page 481

MMEMory:STORe:TRACe:OPTion:RSAMple:TIMStep <TimeStep>

Defines the time step size for resampling time domain traces during export. See `MMEMory:STORe:TRACe:OPTion:RSAMple`.

Alternatively, you can set the number of sampling points (see `MMEMory:STORe:TRACe:OPTion:RSAMple:POINts`).

Parameters:

<TimeStep> Time step size in seconds
The upper limit is the span of the time sweep (see `CALCulate<Chn>:TRANsform:TIME:SPAN`). The lower limit (and default) is this time span divided by N_f-1, where N_f denotes the number of sweep points of the underlying frequency span (see `[SENSe<Ch>:]SWEep:POINts`).
Use `MMEMory:STORe:TRACe:OPTion:RSAMple:PRESet` to set the resampling parameters to their default values.

Options: R&S ZNB-K2

Manual operation: See "Resample/Resample Settings" on page 481

MMEMory:STORe:TRACe:OPTion:SSEParator <Boolean>

This command defines whitespace insertion during Touchstone file export.

If set to ON, separator lines are skipped, i.e. the content parts are no longer separated by blank lines.

Parameters:

<Boolean>

Manual operation: See "Skip Separator Lines" on page 819

MMEMory:STORe:TRACe:OPTion:TABS <Boolean>

This command defines whitespace insertion during Touchstone file export.

If set to ON, columns are separated by tabs rather than spaces.

Parameters:
<Boolean>

Manual operation: See "Use TAB (instead of blanks)" on page 819

MMEMory:STORe:TRACe:OPTion:TRIM <Boolean>

This command defines whitespace insertion during Touchstone file export.

If set to ON, whitespace at the beginning of each line is removed.

Parameters:
<Boolean>

Manual operation: See "Trim Leading Whitespace" on page 819

MMEMory:STORe:TRACe:PORTs <Channel>, <TouchstoneFile>, <Format>[, <ModeImpedance>, <Port>[, <Port>]...]

Generates an `snp`Touchstone file, where `n` is the number of specified ports.

If the specified physical ports comprise at least one balanced port and `MMEMory:STORe:TRACe:OPTion:BALanced` is `ON`, the Touchstone file contains the full set of mixed-mode S-parameters. Otherwise it contains the full set of single-ended S-parameters. Both sets have a size of n^2.

The command fails unless the conditions for Touchstone file export are met; see "Conditions for Touchstone file export" on page 150. Traces are created using the `CALCulate<Ch>:PARameter:SDEFine` command.

Setting parameters:

<Channel> Channel number in the active recall set.

<TouchstoneFile> String parameter to specify the name and directory of the created Touchstone file. The file extension `*.snp` for an `n`-port Touchstone file is mandatory.
If no path is specified, the analyzer searches the current directory, to be queried with `MMEMory:CDIRectory`.

<Format> COMPlex | LINPhase | LOGPhase

COMPlex
Complex values (real and imaginary part)

LINPhase
Linear magnitude and phase

LOGPhase
dB magnitude and phase

<ModeImpedance> CIMPedance | PIMPedance

Impedance renormalization. See "Renormalization of S-parameters" on page 150.

CIMPedance
Normalize to a common target impedance of 50 Ω (=default, if omitted).

PIMPedance
Normalize to the individual port reference impedances.

<Port>, <Port>, ... Physical port numbers

Example: Suppose that a full two-port calibration for ports 1 and 2 and channel 1 has been performed, and that a DUT with two balanced ports is connected. The analyzer measures an arbitrary mixed-mode S-parameter.

```
MMEM:STOR:TRAC:PORT  1, 'Test_CIMP.s2p',
COMPlex, CIMPedance, 1, 2
```

Calculate all single-ended S-parameters, renormalize them to the common target impedance and store them to a two-port Touchstone file.

```
MMEM:STOR:TRAC:PORT  1, 'Test_PIMP.s2p',
COMPlex, PIMPedance, 1, 2
```

Calculate all single-ended S-parameters, renormalize them to the individual port reference impedances and store them to a two-port Touchstone file.

Usage: Setting only

Manual operation: See "Save" on page 481

7.3.12 OUTPut commands

The `OUTPut...` commands control the characteristics of the analyzer's output ports.

OUTPut<Ch>[:STATe] 1300
OUTPut:UPORt:ECBits 1301
OUTPut:UPORt:KEEP 1301
OUTPut<Ch>:UPORt:SEGMent<Seg>:STATe 1301
OUTPut<Ch>:UPORt:SEGMent<Seg>[:VALue] 1302
OUTPut<Ch>:UPORt[:VALue] 1303

OUTPut<Ch>[:STATe] <Boolean>

Turns the internal source power at all ports and the power of all external generators on or off.

Suffix:
<Ch> Channel number. This suffix is ignored; the setting is valid for all channels.

Parameters:
<Boolean> ON | OFF - switch the power on or off.
*RST: ON

Example: OUTP OFF
Turn off the RF source power.

Manual operation: See "RF Off All Channels" on page 531

OUTPut:UPORt:ECBits <Boolean>

Defines the usage of pins 16 to 19 of the USER PORT connector.

Parameters:
<Boolean> ON – channel bits 4 to 7
OFF – drive port 1 to 4
*RST: ON

Example: See OUTPut<Ch>:UPORt[:VALue]

Manual operation: See "Pin 16 - 19" on page 818

OUTPut:UPORt:KEEP <Boolean>

Defines the behavior of the channel bits at sweep end.

Parameters:
<Boolean> **ON (1)**
Channel bits keep their state.
OFF (0)
Channel bits are reset.
*RST: OFF

Manual operation: See "Keep Channel Bits At Sweep End" on page 818

OUTPut<Ch>:UPORt:SEGMent<Seg>:STATe <Boolean>

Enables or disables segment bits for the sweep segments in channel no. <Ch>; see OUTPut<Ch>:UPORt:SEGMent<Seg>[:VALue]. The command is valid for segmented frequency sweep.

Suffix:
<Ch> Channel number.
<Seg> Sweep segment number. This suffix is ignored; the setting is valid for all segments.

Parameters:
<Boolean> ON | OFF - Enables or disables channel bits.
*RST: OFF

Example: See OUTPut<Ch>:UPORt:SEGMent<Seg>[:VALue] on page 1302

Manual operation: See "Optional Columns" on page 550

OUTPut<Ch>:UPORt:SEGMent<Seg>[:VALue] <BinValue>

Sets or queries a sweep segment-dependent four-bit binary value to control four independent output signals at the USER PORT connector (lines 16, 17, 18, 19). The output signals are 3.3 V TTL signals which can be used to differentiate between up to 16 independent analyzer states for each channel. The command is valid for segmented frequency sweeps. It is analogous to the channel-dependent command `OUTPut<Ch>:UPORt[:VALue]`.

The bits for the sweep segments must be enabled explicitly using `OUTPut<Ch>:UPORt:SEGMent:STATe`.

Segment bit definition and activation

The segment bits have the following properties:

- After a `*RST` of the analyzer all segment bits are set to zero; no signal is applied to pins 16 to 19 of the USER PORT connector.
- The value defined with `OUTPut<Ch>:UPORt:SEGMent<Seg>[:VALue]` is assigned to segment no. `<Seg>` in channel no. `<Ch>`.
- The signals at the `USER CONTROL` connector reflect the segments bits of the currently measured segment.
- The signals are switched on as soon as a measurement in a segment with non-zero segment bits is started. They are changed whenever a segment with different segment bits is measured.
- The signals at the USER PORT connector are maintained after the analyzer enters the hold state. This happens in single sweep mode after all sweep sequences have been terminated.

Tip:

You can use the active segment number as a parameter for `OUTPut<Ch>:UPORt:SEGMent<Seg>[:VALue]` and monitor the measurement in up to 16 different segments per channel at the USER PORT connector; see example below. You can also use the USER PORT output signals as segment-dependent trigger signals for external devices. Use `CONTrol:AUXiliary:C[:DATA]` to transfer the four bit value in decimal representation.

Suffix:

<Ch> Channel number.

<Seg> Sweep segment number

Parameters:

<BinValue> Binary value. The transferred values correspond to the following states of the USER CONTROL connector:
#B0000 - no signal at any of the four pins 16, 17, 18, 19
#B0001 - output signal at pin 16
#B0010 - output signal at pin 17
#B0011 - output signal at pin 16 and 17
...
#B1111 - output signal at pin 16, 17, 18 and 19

Range: #B0000 to #B1111 (for setting command), 0 to 15 (query)
*RST: #B0000 (0)

Example:
```
*RST; :SEGM:INS 1MHZ, 1.5MHZ, 111, -21DBM, 0.5S, 0, 10KHZ
```
Create a sweep segment no. 1 with a sweep range between 1.0 MHz and 1.5 MHz.
```
SWE:TYPE SEGM
```
Set the segmented frequency sweep active.
```
OUTP:UPOR:SEGM:STAT ON
```
Enable segment bits.
```
OUTP:UPOR:SEGM1 #B0001
```
Assign the segment bit value #B0001 to segment no. 1. The output signal at pin 16 is switched on while the first segment is measured.
```
SEGM2:ADD
```
Create a second sweep segment. The frequency range of the second segment will be between 1.5 MHz and the maximum frequency of the analyzer.
```
OUTP:UPOR:SEGM2 #B0010
```
Assign the segment bit value #B0010 to segment no. 2. While the analyzer measures the second segment, the output signal changes from pin 16 to pin 17.

Manual operation: See "Optional Columns" on page 550

OUTPut<Ch>:UPORt[:VALue] <BinValue>

Sets or queries a channel-dependent eight-bit binary value to control eight independent output signals at the USER PORT connector (lines 8, 9, 10, 11 and lines 16, 17, 18, 19). The output signals are 3.3 V TTL signals which can be used to differentiate between up to 255 independent analyzer states. `OUTPut<Ch>:UPORt[:VALue]` itself does not change the analyzer state.

Channel bit definition and activation

The channel bits have the following properties:

- After a `*RST` of the analyzer all channel bits (including the value for the active, sweeping channel no. 1) are set to zero; no signal is applied to pins 8 to 11 and 16 to 19 of the USER PORT connector.
- The value defined with `OUTPut<Ch>:UPORt[:VALue]` is assigned to channel no. `<Ch>`.
- The signals at the USER PORT connector reflect the channel bits of the **measuring** channel, i.e. the channel for which the analyzer performs a sweep. This channel is not necessarily identical with the active channel.
- The signals are switched on as soon as a measurement (sweep) in a channel with non-zero channel bits is started. They are changed whenever a channel with different channel bits becomes the measuring channel.

- The signals at the USER PORT connector are maintained after the analyzer enters the hold state. This happens if all channels use single sweep mode and if all sweep sequences have been terminated.
- Pins 16 to 19 may be reserved for monitoring the drive ports 1 to 4 of the analyzer (`OUTPut<Ch>:UPORt:ECBits OFF`) . This leaves up to 16 different monitored channel states.

Tip: You can use the active channel number as a parameter for `OUTPut<Ch>:UPORt[:VALue]` and monitor the activity of up to 255 different channels at the USER PORT connector; see example below. You can also use the USER PORT output signals as channel-dependent (or drive port-dependent) trigger signals for external devices. Furthermore you can use `CONTrol:AUXiliary:C[:DATA]` to transfer the eight bit value in decimal representation.

Suffix:
<Ch> Channel number

Parameters:
<BinValue> Binary value. The values correspond to the following states of the USER PORT connector:
#B00000000 - no signal at any of the eight pins 8, 9, 10, 11, 16, 17, 18, 19
#B00000001 - output signal at pin 8
#B00000010 - output signal at pin 9
#B00000011 - output signal at pins 8 and 9
...
#B11111111 - output signal at pins 8, 9, 10, 11, 16, 17, 18, 19

Range: #B00000000 to #B11111111 (for setting command), 0 to 255 (query)
*RST: #B00000000 (0)

Example:
```
*RST; :OUTP1:UPOR #B00000001
```
Assign the channel bit value #B00000001 to the active channel no. 1. The analyzer performs a measurement in channel no. 1, therefore the output signal at pin 8 is switched on.
```
CONF:CHAN2:STAT ON; OUTP2:UPOR #B00000010
```
Create channel no. 2, causing it to become the active channel, and assign the channel bit value #B00000010. The analyzer performs no measurement in channel no. 2, therefore the output signal is not changed.
```
CALC2:PAR:SDEF 'Ch2Tr1', 'S11'
```
Create a trace named 'Ch2Tr1' and assign it to channel 2. While the analyzer measures in channel 2, the output signal changes from pin 8 to pin 9.
```
OUTP:UPOR:ECB OFF
```
Reserve pin 16 to 19 for monitoring the drive ports of the analyzer.

Manual operation: See "Channel Bits (Decimal)" on page 817

7.3.13 PROGram commands

The PROGram... commands control external application programs that can be run on the analyzer.

PROGram[:SELected]:EXECute 1305
PROGram[:SELected]:INIMessage 1306
PROGram[:SELected]:INIParameter 1306
PROGram[:SELected]:NAME 1307
PROGram[:SELected]:RETVal? 1308
PROGram[:SELected]:WAIT 1308

PROGram[:SELected]:EXECute <AppName>

Starts an application process or open a file using an application available on the analyzer.

Use the comman sequence PROGram[:SELected]:WAIT? ; PROGram[:SELected]:RETVal? to query the return value (see example below).

Note: It is not possible to run several programs simultaneously. If the command PROGram[:SELected]:EXECute ... is sent while a previously started program is still executed, the analyzer generates a SCPI error -100,"Command error...".

Tip: Executing batch files; command prompt

When executing batch scripts or other DOS applications, the analyzer does not display any DOS windows; the screen is left for the vector network analyzer (VNA) application. The same applies to the Windows NT command prompt (cmd.exe). To access the command prompt, proceed as follows:

- Create a batch file (e.g. Start_cmd.bat) containing the command line start cmd.exe and store the file to C:\Winnt\system32.
- Execute the batch file: PROG:EXEC 'C:\winnt\system32\Start_cmd.bat'

The command prompt window is displayed in front of the VNA application. You can also open several command prompt windows simultaneously.

Setting parameters:

<AppName> String variable containing the name and path of an application program to be executed or of a file to be opened. The path can be defined as an absolute path (e.g. 'c:\...') or relative to the current directory (MMEMory:CDIRectory).
Blanks in the <AppName> can be used to separate the application name from (optional) parameters.

Example: `PROGram:SELected:NAME PROG`
Select general program execution.
`PROGram:SELected:EXECute 'Exit42.bat'`
Run batch script `Exit42.bat`.
`PROGram:SELected:WAIT?`
Lock command execution and manual control of the analyzer until the batch job has finished. This is required for
`PROGram:SELected:RETVal?`
Get the return value. The answer is ... 42.

Usage: Setting only

PROGram[:SELected]:INIMessage <IniFile>[, <SendValue>]

Writes a message `<SendValue>` into the preferences (`*.ini`) file specified by `<IniFile>`. The message is entered into the `[MESSAGE]` section using the fixed key `Send`; the value for the fixed key `Receive` is set to an empty string.

The query reads the value associated with the fixed key `Receive` from the `[MESSAGE]` section of the preferences file specified by `<IniFile>`. If no value exists for that key, the query returns an empty string.

Both commands can be used to establish a simple file-based two-way communication mechanism to an external application launched by `PROGram[:SELected]:EXECute`; see example.

Parameters:

<IniFile> Name and path of the `*.ini` file. The `*.ini` extension may be omitted as it is created automatically by the command. The specified path/directory must exist. If the `*.ini` file does not exist, it is created.

<SendValue> Value for the fixed key `Send`.

Example: `PROG:INIM 'c:\preferences\myapp', 'this is a message'`
Write the string `this is a message` into the file `c:\preferences\myapp.ini`. The contents of the file look like:
`[MESSAGE] Send="this is a message" Receive=`
Suppose the external program writes the string `this is a response` to the `Receive` key (and possibly deletes the contents of the `Send` key.
`PROG:INIM? 'c:\preferences\myapp'`
Query the value of the key `Receive` in the `*.ini` file. The response is `"this is a response"`.

PROGram[:SELected]:INIParameter <IniFile>{ , <Key>,<Value> | '<Value>'}

Defines and writes one or several key/value pairs into the preferences file (`*.ini`) specified by `<file_path>`. The information is entered into the `[PARAMETER]` section.

This command can be used to supply information to an external application launched by :PROGram[:SELected]:EXECute.

The query must be sent with a single <Key> value. It reads the value associated with the key from the [PARAMETER] section of the preferences file specified by <file_path>. If the key/value pair does not exist, the query returns an empty string.

Parameters:

<IniFile> Name and path of the *.ini file. The *.ini extension may be omitted as it is created automatically by the command. The specified path/directory must exist. If the *.ini file does not exist, it is created.

<Key> Key for the key/value pair(s).

<Value> String or numeric value for the key/value pair(s). If a string parameter is supplied, it has to be enclosed in single or double quotes.

Example:

```
PROG:INIP 'c:\preferences\myapp',
'myparameter', 'myvalue', 'startf', 123.05
```

Write two key/value pairs into the file c:\preferences\myapp.ini. The contents of the file look like:

```
[PARAMETER]
myparameter="myvalue"
startf="123.05"
PROG:INIP? 'c:\preferences\myapp',
'myparameter'
```

Query the value of the key myparameter in the *.ini file. The response is "myvalue".

PROGram[:SELected]:NAME <Program>

Selects the application to be run on the analyzer. At present, only the general parameter PROG is available. This means that PROGram[:SELected]:EXECute can start any program.

Tip: Use this command in order to avoid problems should the default value change in future firmware versions.

Parameters:

<Program> PROG

Any program running under Windows or any file that can be opened with an application program available on the analyzer.

*RST: PROG

Example: See PROGram[:SELected]:EXECute

PROGram[:SELected]:RETVal?

Queries the return value of an application or process started via `PROGram[:SELected]:EXECute`.

This will only be successful if preceded by a `PROGram[:SELected]:WAIT?` query (see `PROGram[:SELected]:WAIT` on page 1308).

Example: See `PROGram[:SELected]:EXECute`

Usage: Query only

PROGram[:SELected]:WAIT

Locks command execution from the current controller program while a program started via `PROGram[:SELected]:EXECute` is running. The analyzer does not execute any further commands or queries until the program is stopped or paused.

Use `PROGram[:SELected]:WAIT?` before trying to retrieve the return value of the executed program (`PROGram[:SELected]:RETVal?`).

Example: See `PROGram[:SELected]:EXECute`

7.3.14 [SENSe:] commands

The `[SENSe:]...` commands affect the receiver settings of the R&S ZNB.

7.3.14.1 [SENSe:]AVERage...

The `[SENSe:]AVERage...` commands set sweep averaging parameters. The sweep average is a noise-reduction technique which consists of calculating each measurement point as an average of the same measurement point over several consecutive sweeps.

In contrast to the sweep count (for single sweep mode, `[SENSe<Ch>:]SWEep:COUNt`), averaging is always channel-specific. Both features are independent from each other.

[SENSe<Ch>:]AVERage[:STATe] 1308
[SENSe<Ch>:]AVERage:CLEar 1309
[SENSe<Ch>:]AVERage:COUNt 1309
[SENSe<Ch>:]AVERage:MODE 1309

[SENSe<Ch>:]AVERage[:STATe] <Boolean>

Enable or disable the sweep average.

Suffix:
<Ch> Channel number

Parameters:

<Boolean> ON | OFF - enables or disables the automatic calculation of the sweep average over the specified number of sweeps ([SENSe<Ch>:]AVERage:COUNt).

*RST: OFF

Example: See [SENSe<Ch>:]AVERage:CLEar

Manual operation: See "Factor / On / Reset" on page 535

[SENSe<Ch>:]AVERage:CLEar

Starts a new average cycle, clearing all previous results and thus eliminating their effect on the new cycle.

Suffix:

<Ch> Channel number

Example: SENS1:AVER:COUN 15; :AVER ON
Set the average factor for channel 1 to 15 (the mnemonic SENS1 can be omitted) and enable the sweep average.
AVER:COUN 5; CLE
Reduce the average factor and restart the average.

Usage: Event

Manual operation: See "Factor / On / Reset" on page 535

[SENSe<Ch>:]AVERage:COUNt <AverageFactor>

Defines the number of consecutive sweeps to be combined for the sweep average ("Factor").

Suffix:

<Ch> Channel number

Parameters:

<AverageFactor> Sweep average factor

Range: 1 to 1000
*RST: 10

Example: See [SENSe<Ch>:]AVERage:CLEar

Manual operation: See "Factor / On / Reset" on page 535

[SENSe<Ch>:]AVERage:MODE <Mode>

Suffix:

<Ch>

Parameters:

<Mode> AUTO | FLATten | REDuce | MOVing

AUTO
Automatic selection between REDuce and FLATten mode, depending on the trace format.

FLATten
Cumulative moving averages of the (linear) magnitude and phase values, provides the most effective noise suppression for the "dB Mag", "Phase", "Unwr Phase", and "Lin Mag" formats.

REDuce
Cumulative moving averages of the real and imaginary parts of each measurement result, provides the most effective noise suppression for the "Real" and "Imag" formats and for complex trace formats.

MOVing
Simple moving averages of the real and imaginary parts of each measurement result; similar to REDuce, but with finite history.

Manual operation: See "Mode" on page 535

7.3.14.2 [SENSe:]BANDwidth...

The `[SENSe:]BANDwidth...` commands set the bandwidth of the IF filter (measurement bandwidth). The forms `BANDwidth` and `BWIDth` are equivalent.

[SENSe<Ch>:]BANDwidth[:RESolution] 1310
[SENSe<Ch>:]BWIDth[:RESolution] 1310
[SENSe<Ch>:]BANDwidth[:RESolution]:SELect 1311
[SENSe<Ch>:]BWIDth[:RESolution]:SELect 1311

[SENSe<Ch>:]BANDwidth[:RESolution] <ResBandw>
[SENSe<Ch>:]BWIDth[:RESolution] <IF Bandwidth>

Defines the IF bandwidth of the analyzer (measurement bandwidth). Values between 1 Hz and 1 MHz can be set. Option R&S ZNB-K17 enables bandwidths up to 10 MHz (see Section 4.7.4, "Receiver bandwidth 10 MHz", on page 243).

Suffix:
<Ch> Channel number

Parameters:
<IF Bandwidth> Bandwidths can be set in 1 – 1.5 – 2 – 3 – 5 – 7 steps. The analyzer rounds up any entered value between these steps and rounds down values exceeding the maximum bandwidth.

Range: See above
*RST: 10 kHz
Default unit: Hz

Example:
```
BAND 1.1
```
Set an IF bandwidth of approx. 1.1 Hz for channel 1.
```
BAND?
```
The analyzer returns the rounded bandwidth of 1.5 Hz.

Manual operation: See "Meas Bandwidth" on page 340

[SENSe<Ch>:]BANDwidth[:RESolution]:SELect <Selectivity>
[SENSe<Ch>:]BWIDth[:RESolution]:SELect <Selectivity>

Defines the selectivity of the IF filter for an unsegmented sweep. The value is also used for all segments of a segmented sweep, provided that separate selectivity setting is disabled (`[SENSe<Ch>:]SEGMent<Seg>:BWIDth[:RESolution]:SELect:CONTrol OFF`).

Suffix:
<Ch> Channel number

Parameters:
<Selectivity> NORMal | MEDium | HIGH

NORMal – IF filter with normal selectivity and short settling time.
MEDium – IF filter with steeper edges and longer settling time.
HIGH – IF filter with highest selectivity but longest settling time.

*RST: NORMal

Example: See `[SENSe<Ch>:]SEGMent<Seg>:BWIDth[:RESolution]:SELect:CONTrol`

Manual operation: See "Selectivity" on page 341

7.3.14.3 [SENSe:]CDLL...

Adds, removes and configures custom (external) DLLs.

These commands allow you to add/remove external DLLs to/from the firmware. It gives access to their configuration, tasks, and traces.

[SENSe:]CDLL:ADD 1311
[SENSe:]CDLL:LIST? 1312
[SENSe:]CDLL:LIST:TASK? 1312
[SENSe<Ch>:]CDLL:PERManent:ADDitional 1312
[SENSe<Ch>:]CDLL:PERManent[:STATe] 1313
[SENSe<Ch>:]CDLL:PERManent:TASK 1313
[SENSe:]CDLL:REMove 1313
[SENSe<Ch>:]CDLL[:STATe] 1314
[SENSe<Ch>:]CDLL:TASK:ADDRess 1314

[SENSe:]CDLL:ADD <DllPath>

Tells the analyzer firmware to load a custom (external) DLL.

The DLL and its accompanying files must be packed in a zip archive. It will be loaded on every subsequent firmware start, until it is removed using `[SENSe:]CDLL:REMove` .

Note that adding/removing custom DLLs is only possible if the firmware is run as administrator.

Setting parameters:
<DllPath> Full path to the zip archive including the DLL (at the top-tevel).

Usage: Setting only

Manual operation: See "Add / Remove" on page 435

[SENSe:]CDLL:LIST?

Queries the loaded custom (external) DLL files.

The return value is a single string, comprising a comma-separated list of DLL names, without the `dll` extension.

Use `[SENSe:]CDLL:ADD`/`[SENSe:]CDLL:REMove` to load/unload a custom DLL..

Usage: Query only

Manual operation: See "Loaded DLLs table" on page 434

[SENSe:]CDLL:LIST:TASK? <DllName>

Lists the tasks implemented by the custom (external) DLL <DllName>.

Query parameters:
<DllName> Name of a loaded DLL (see `[SENSe:]CDLL:LIST?`)
Lowercase string, without `dll` extension

Usage: Query only

[SENSe<Ch>:]CDLL:PERManent:ADDitional <DllName>, <Additional Input>
[SENSe<Ch>:]CDLL:PERManent:ADDitional? <DllName>

Defines additional input for the task to be run, if custom DLL <DllName> is set to permanent mode in channel <Ch> (`[SENSe<Ch>:]CDLL:PERManent[:STATe] '<'DllName>', ON`).

Use `[SENSe<Ch>:]CDLL:PERManent:TASK` to select the type of task to be run.

Suffix:
<Ch> Channel number

Parameters:
<Additional Input> String value, with DLL-specific syntax and semantics. See the docs of your DLL for details.

Parameters for setting and query:
<DllName> Name of a loaded DLL (see `[SENSe:]CDLL:LIST?`)
Lowercase string, without `dll` extension

Manual operation: See "Additional Input" on page 435

[SENSe<Ch>:]CDLL:PERManent[:STATe] <DllName>, <Boolean>
[SENSe<Ch>:]CDLL:PERManent[:STATe]? <DllName>

Defines whether DLL <DllName> shall operate in permanent mode in channel <Ch>.

Use `[SENSe<Ch>:]CDLL:PERManent:TASK` and `[SENSe<Ch>:]CDLL:PERManent:ADDitional` to configure the permanent task to be run.

Suffix:
<Ch> Channel number

Parameters:
<Boolean> **ON (1)**
Permanent mode enabled
OFF (0)
Permanent mode disabled

Parameters for setting and query:
<DllName> Name of a loaded DLL (see `[SENSe:]CDLL:LIST?`)
Lowercase string, without `dll` extension

Manual operation: See "Permanent" on page 435

[SENSe<Ch>:]CDLL:PERManent:TASK <DllName>, <Task>
[SENSe<Ch>:]CDLL:PERManent:TASK? <DllName>

Defines the type of task to be run, if custom DLL <DllName> is set to permanent mode in channel <Ch> (`[SENSe<Ch>:]CDLL:PERManent[:STATe] '<'DllName>', ON`).

Use `[SENSe<Ch>:]CDLL:PERManent:ADDitional` to further specify the task.

Suffix:
<Ch> Channel number

Parameters:
<Task> Name of the task to be run permanently.

Parameters for setting and query:
<DllName> Name of a loaded DLL (see `[SENSe:]CDLL:LIST?`)
Lowercase string, without `dll` extension

Manual operation: See "Task Type" on page 435

[SENSe:]CDLL:REMove <DllName>

Tells the analyzer firmware to remove a custom (external) DLL that was previously added using `[SENSe:]CDLL:ADD`.

Note that adding/removing custom DLLs is only possible if the firmware is run as administrator.

Setting parameters:

<DllName> Name of a loaded DLL (see `[SENSe:]CDLL:LIST?`)
Lowercase string, without `dll` extension

Usage: Setting only

Manual operation: See "Add / Remove" on page 435

[SENSe<Ch>:]CDLL[:STATe] <DllName>, <Boolean>
[SENSe<Ch>:]CDLL[:STATe]? <DllName>

Defines the state of (external) DLL `<DllName>.dll` in channel <Ch>.

An active DLL can provide traces or operate in permanent mode (see `[SENSe<Ch>:]CDLL:PERManent[:STATe]`).

Suffix:

<Ch> Channel number

Parameters:

<Boolean> **ON (1)**
DLL is active

OFF (0)
DLL is inactive

Parameters for setting and query:

<DllName> Name of a loaded DLL (see `[SENSe:]CDLL:LIST?`)
Lowercase string, without `dll` extension

Manual operation: See "Active" on page 435

[SENSe<Ch>:]CDLL:TASK:ADDRess <DllName>, <Task>, <Address>
[SENSe<Ch>:]CDLL:TASK:ADDRess? <DllName>, <Task>

Defines the address of an external device the external DLL <DllName> requires to complete task type <Task> in channel <Ch>.

Suffix:

<Ch> Channel number

Parameters:

<Address> String value with DLL-specific syntax and semantics. See the docs of your DLL for details.

Parameters for setting and query:

<DllName> Name of a loaded DLL (see `[SENSe:]CDLL:LIST?`)
Lowercase string, without `dll` extension

<Task> Task type (see `[SENSe:]CDLL:LIST:TASK?`)

Manual operation: See "Configure Device Adress" on page 436

7.3.14.4 [SENSe:]CORRection:CKIT...

The `[SENSe:]CORRection:CKIT...` commands deal with calibration kits and cal kit data. The calibration kits are distinguished by their names (<CalkitName>), the optional labels (<label>) can be used to carry information about the calibration standard.

To handle several identical calibration kits with different serial numbers, use the commands of Section 7.3.14.5, "[SENSe:]CORRection:CKIT... with labels", on page 1324.

[SENSe:]CORRection:CKIT:<StandardType> 1315
[SENSe:]CORRection:CKIT:ADD 1319
[SENSe:]CORRection:CKIT:CATalog? 1319
[SENSe:]CORRection:CKIT:COPY 1320
[SENSe:]CORRection:CKIT:DELete 1321
[SENSe:]CORRection:CKIT:DMODe 1321
[SENSe:]CORRection:CKIT:LABel 1321
[SENSe:]CORRection:CKIT:SELect 1322
[SENSe:]CORRection:CKIT:<ConnType>:SELect 1322
[SENSe:]CORRection:CKIT:STANdard:CATalog? 1323
[SENSe:]CORRection:CKIT:STANdard:DATA? 1323
[SENSe:]CORRection:CKIT:THRough:SELect 1324

[SENSe:]CORRection:CKIT:<StandardType> <ConnType>, <CalKitName>, <StandardLabel>, <MinFreq>, <MaxFreq>[, <DelayParam>, <Loss>, <Z0>[, <C0>, <C1>, <C2>, <C3>, <L0>, <L1>, <L2>, <L3>[, OPEN | SHORt | MATCh | <Resistance>[, <Port1>[, <Port2>]]]]]

[SENSe:]CORRection:CKIT:<StandardType>? <ConnType>, <CalKitName>[, <Port1>[, <Port2>]]

Defines the parameters of a (possibly non-ideal) 1 port or 2-port calibration standard `<StandardType>` within a particular cal kit. Depending on the standard type, only a subset of the parameters is used; see Table 7-13

Note

As the query does not reflect the delay mode specified using `[SENSe:]CORRection:CKIT:DMODe`, it is deprecated and only available for backward compatibility reasons. Use `[SENSe:]CORRection:CKIT:STANdard:DATA??` instead.

Suffix:

<StandardType> <string>
Standard type
For one-port standards, the first character denotes the gender, for transmission standards the first two characters denote the genders on both ends, e.g. `MOPen` for a male Open standard or `FFTHrough` for a Through standard with female connectors.
For a complete list of standards, refer to Table 7-15.

Parameters:

<StandardLabel> Additional string label for the standard, typically the standard's serial number

CalStandardProps — Parameters <MinFreq>, ..., OPEN | SHORt | MATCh | <Resistance> define the properties of the calibration standard. See Parameter list.
Note: Sliding Match and Attenuation standards have only 2 parameters (<MinFreq>,<MaxFreq>). Through and Line standards only have 5 parameters (<MinFreq>,...,<Z0>).

<Port1>, <Port2> — Optional port restriction.
For a one-port standard, the validity of the characterization can be restricted to a single port. For a two-port standard, it can be restricted to a port pair (specified using ascending port numbers).
Note: with a port restriction, the defined standard becomes sexless. Hence, for each port (pair) there can be only one standard of a given type, i.e. `:SENSe:CORRection:CKIT:MOP <ParameterList>,1` and `:SENSe:CORRection:CKIT:FOP <ParameterList>,1` overwrite each other.

Parameters for setting and query:

<ConnType>, <CalkitName> — String parameters uniquely identifying the cal kit to which the standard belongs.
Note: If the specified cal kit does not exist, it is created with the specified calibration standard.

Example:
```
CORR:CKIT:FOP 'N 50 Ohm','New Kit 1', '',
0,4000000000,0,0,50,99,-2.3,0.22,0,0,0,0,0
```
Define the properties of the Open (f) standard for a N 50 Ω connector type in cal kit "New Kit 1".
```
CORR:CKIT:FOP? 'N 50 Ohm'
```
Query the properties of the Open (f) standard for a N 50 Ω connector type in the active cal kit.
```
CORR:CKIT:FOP? 'N 50 Ohm','New Kit 1'
```
Query the properties of the Open (f) standard for a N 50 Ω connector type in cal kit "New Kit 1".

Manual operation: See "One Port Standards / Two Port Standards" on page 640

Table 7-13: Set command parameters

	One port standards					Two port standards		
	Open	(Offset) Short	Match	Reflect	Sliding match	Symmetric network	Through, Line	Attenuation
<ConnType>	mandatory							
<CalKitName>	mandatory (and must not be an empty string)							
<StandardLabel>	mandatory (but can be an empty string)							
<MinFreq>, <MaxFreq>	mandatory							

<Delay-Param>, <Loss>, <Z0>	mandatory	mandatory	mandatory	mandatory	not used	mandatory	not used
<C0>, <C1>, <C2>, <C3>	mandatory	mandatory*	mandatory*	mandatory**		mandatory**	not used
<L0>, <L1>, <L2>, <L3>	mandatory*	mandatory	mandatory*	mandatory**		mandatory**	
OPEN | SHORt | MATCh | <Resist-ance>	optional***	optional***	optional***	mandatory: OPEN | SHORt | <Resist-ance>		mandatory: OPEN | SHORt | <Resist-ance>	
<Port1>[, <Port2>]	optional: <Port 1>					optional: <Port 1>,<Port 2>	

* values are ignored during calibration

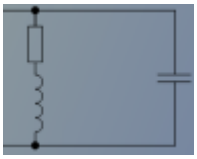
** if OPEN is selected the residual inductance (L0,...,L3) are during calibration; if SHORt is selected the fringing capacitance (C0,...,C3) are during calibration

*** must be provided if the definition is restricted to a single port. Values are ignored during calibration

The parameters in the `[SENSe<Ch>:]CORRection:CKIT:<StandardType>`, `[SENSe<Ch>:]CORRection:CKIT:<StandardType>:WLABels`, and `[SENSe<Ch>:]CORRection:CKIT:<ConnectorType>:<StandardType>` commands have the following meaning:

Table 7-14: Parameter list

Parameter	Meaning	Comment/Unit
'<ConnType>'	Name of the connector type. Use `SENSe1:CORRection:CONNection:CATalog?` to query connector names.	String parameter
'<CalKitName>'	Name of the calibration kit. Use `[SENSe:]CORRection:CKIT:CATalog?` to query cal kit names.	String parameter
'<CalKitLabel>'	Label (e.g. the serial number) of the calibration kit; for `...WLABel` command only	String parameter
'<StandardLabel>'	Label (e.g. the serial number) of the standard	String parameter
<MinFreq>, <MaxFreq>	Min./max. frequency for which the circuit model is valid	Default unit is Hz
<DelayParam>	Depending on the cal kit's model type (selected using `[SENSe:]CORRection:CKIT:DMODe`), this parameter is interpreted as: • delay [s] for Keysight modeling • el. length [m] for ZVR compatible modeling Cal kits that are not created/modified using `[SENSe:]CORRection:CKIT:DMODe` use the ZVR compatible modeling.	
<Loss>	Loss (offset parameter) of the standard	Must be specified without unit (implicit unit is dB)

Parameter	Meaning	Comment/Unit
<Z0>	Reference impedance (no unit)	Must be specified without unit (implicit unit is Ω)
<C0>,...,<C3>	Polynomial coefficients for the fringing capacitance of the standard (load parameter)	Must be specified without unit: implicit unit of <C*i*> is fF / (GHz)^i
<L0>,...,<L3>	Polynomial coefficients for the residual inductance of the standard (load parameter)	Must be specified without unit: implicit unit of <L*i*> is pH / (GHz)^i
OPEN | SHORt | MATCh | <Resistance>	A load circuit model generally consists of a capacitance (modeled by <C0>,...,<C3>), connected in parallel to an inductance (modeled by <L0>,...,<L3>) and a resistance, both connected in series. OPEN | SHORt | MATCh indicates a simplified modeling as an Open or Short or Match standard. • OPEN: the resistance is infinite so that the standard behaves like a capacitor (no inductance) • SHORt: the resistance is zero so that the standard behaves like an inductance (no capacitance) • MATCh: the standard behaves like a match (no inductance, no capacitance, resistance Z0) <Resistance> indicates the general load circuit model.	 Character data | numeric value
<Port1>, <Port2>	Optional port restriction: one port number for one-port standards, two port numbers for two-port standards	Integer values

The different standard types are defined by the following parameters. Port restrictions are indicated in brackets:

Table 7-15: Standard types and their parameters

<std_type>	Meaning
MOPen | FOPen	Open: male (m) or female (f)
MSHort | FSHort	Short: m or f
OSHort[<1|2|3>] | MOSHort[<1|2|3>] | FOSHort[<1|2|3>]	Offset short: sexless, m or f (three standards each) For user-defined connector types only. Suffix 1 can be omitted.
MMTCh | FMTCh	Match: m or f
MSMatch | FSMatch	Sliding match: m or f
MREFlect | FREFlect	Reflect: m or f
MMTHrough | MFTHrough | FFTHrough	Through: m-m or m-f or f-f
MMLine[<1,...,6>] | MFLine[<1,...,6>] | FFLine[<1,...,6>] MMLine[<1,...,6>](P2P3) ...	Line: m-m or m-f or f-f (three standards each) Suffix 1 can be omitted.
MMATten | MFATten | FFATten	Attenuation: m-m or m-f or f-f
MMSNetwork | MFSNetwork | FFSNetwork	Symmetric network: m-m or m-f or f-f

[SENSe:]CORRection:CKIT:ADD <ConnectorName>, <KitName>, <KitLabel>, <DelayMode>

Defines a new calibration kit for connector <ConnectorName>.

Setting parameters:

<ConnectorName> Connector name (ASCII string)
Use `[SENSe:]CORRection:CONNection:CATalog?` to get a list of the existing connector names.

<KitName> ASCII string, defining the name of the calibration kit.

<KitLabel> ASCII string, defining the label of the calibration kit; typically the kit's serial number.
Mandatory, but can be empty.

<DelayMode> DELay | ELENgth

DELay
Keysight modeling

ELENgth
ZVR-compatible modeling

Usage: Setting only

Manual operation: See "Add / Copy / Delete / Standards" on page 638

Using "Ohm" in connector names, cal kit names, and cal kit labels

Each occurrence of the string "Ohm" (case-insensitive) in <ConnectorName>, <KitName>, or <KitLabel> is converted to the symbol "Ω" at the VNA **GUI**. Conversely, if you define a cal connector or cal kit at the GUI, then each "Ω" occurring in their name or label is converted to "Ohm" at the remote interface.

[SENSe:]CORRection:CKIT:CATalog? [<ConnectorName>]

Queries all cal kits that are available for cal connector type <ConnectorName>, or for all cal connector types (if <ConnectorName> is omitted).

The return value is a string, containing a comma-separated list `'<kit1_name> (<kit1_label>),<kit2_name> (<kit2_label>), ...'`, where the kit label part in parentheses is only appended, if a label is assigned to the respective kit.

Query parameters:

<ConnectorName> Name of a connector type (ASCII string, optional).
Use `[SENSe:]CORRection:CONNection:CATalog?` to query connector names.

Example: `CORRection:CONNection:CATalog?`
Query connector types. Possible response: `'N 50 Ohm,N 75 Ohm,7 mm,3.5 mm,2.92 mm,2.4 mm,1.85 mm,7-16,Type F (75),BNC 50 Ohm,BNC 75 Ohm,SMA,4.3-10,User Conn 1'`
`CORRection:CKIT:CATalog? 'N 50 Ohm'`
Query cal kits for connector type 'N 50 Ohm' ("N 50 Ω" at the GUI).
Possible response: `'N 50 Ohm Ideal Kit,3653,85054D,ZV-Z121,ZCAN 50 Ohm,ZV-Z21 typical,85032B/E,85032F,85054B,New Kit 1'`.
We assume that a cal kit `New Kit 1` with label `2012-05-25` was created before.
`CORRection:CKIT:LCATalog? 'N 50 Ohm'`
Query cal kits for N (50 Ω) connector types with their labels.
Possible response: `'N 50 Ohm Ideal Kit,,3653,,85054D,,ZV-Z121,,ZCAN 50 Ohm,,ZV-Z21 typical,,85032B/E,,85032F,,85054B,,New Kit 1,2012-05-25'`. A sequence of two commas means that the preceding cal kit has no label assigned.
`CORRection:CKIT:STANdard:CATalog? '85032B/E'`
Query standards in cal kit named 85032B/E. Possible response: `'MOP,FOP,MSH,FSH,MMMT,FMMT,MFTH'`
`CORRection:CKIT:STANdard:LCATalog? 'New Kit 1', '2012-05-25'`
Query standards in user cal kit named `New Kit 1` labeled `2012-05-25`. Possible response: `'MOP,FOP,MSH(P1),FSH,MMMT,FMMT,MSM,FSM,MREF,FREF, MOSHORT2,FOSHORT2,MOSHORT3,FOSHORT3,MMTH,MMTH(P2P3) ...'` (P1 etc. denote restricted port assignments).

Usage: Query only

Manual operation: See "Available Cal Kits" on page 638

[SENSe:]CORRection:CKIT:COPY <KitName>[, <KitLabel>]

Copies an existing calibration kit, identified by its name (and label).

Setting parameters:

<KitName> String parameter specifying the name of the calibration kit to be copied.

<KitLabel> String parameter specifying the label of the calibration kit to be copied.
Assumed empty, if omitted.

Usage: Setting only

Manual operation: See "Add / Copy / Delete / Standards" on page 638

[SENSe:]CORRection:CKIT:DELete <CalKitName>

Deletes an imported or user-defined cal kit.

Note: It is not possible to modify or store predefined or ideal kits.

Setting parameters:

<CalKitName> String parameter containing an imported or user-defined calibration kit available on the analyzer.

Example: See `[SENSe:]CORRection:CKIT:LABel`

Usage: Setting only

Manual operation: See "Add / Copy / Delete / Standards" on page 638

[SENSe:]CORRection:CKIT:DMODe <ConnectorName>, <KitName>, <KitLabel>, <Mode>
[SENSe:]CORRection:CKIT:DMODe? <ConnectorName>, <KitName>, <KitLabel>

Sets/gets the delay mode for the related cal kit (identified by connector type, name and label), i.e allows to toggle between ZVR compatible and Keysight modelling (see "Offset Parameters" on page 642). Subsequent standard definitions interpret the specified <DelayParam> accordingly.

In "set direction", if a cal kit with the given connector type, name and label is not available on the analyzer, it is created automatically.

Cal kits that are not created/modified using `[SENSe:]CORRection:CKIT:DMODe` use the ZVR compatible modelling.

Parameters:

<ConnectorName> Name of the connector type.
Use `[SENSe:]CORRection:CONNection:CATalog?` to query connector names.

<KitName>, <KitLabel> Name and label of the calibration kit.
Use `[SENSe:]CORRection:CKIT:LCATalog? <ConnectorName>` to query for these cal kits.

Setting parameters:

<Mode> DELay | ELENgth

DELay – Keysight modelling
ELENgth – ZVR compatible modelling

Manual operation: See "Add / Copy / Delete / Standards" on page 638

[SENSe:]CORRection:CKIT:LABel <CalKitName>[, <KitLabel>]

Assigns a label to an imported or user-defined calibration kit.

Parameters:

<CalKitName> String parameter containing an imported or user-defined calibration kit available on the analyzer.

<KitLabel> String parameter containing the calibration kit label.

Example:
```
CORR:CKIT:FOP 'N 50 Ohm','New Kit 1', '',
0,4000000000,0,0,50,99,-2.3,0.22,0,0,0,0,0
```
Create a new cal kit "New Kit 1" and assign an open (f) standard for the N 50 Ω connector type with specific properties.
```
CORR:CKIT:LAB 'New Kit 1', 'Test kit created
today'
```
Label the previously created kit.
```
CORR:CKIT:LAB? 'New Kit 1'
```
Check the label.
```
CORR:CKIT:DEL 'New Kit 1'
```
Delete the kit.

[SENSe:]CORRection:CKIT:SELect <ConnectorName>[, <CalKitName>]

Sets or queries the calibration kit to be used for connector type <ConnectorName>.

Tip: This command selects a cal kit by its name <CalKitName> – which can be ambiguous. Use `[SENSe:]CORRection:CKIT:LSELect` to select a cal kit by its name *and* label.

Parameters:

<ConnectorName> Connector type name (ASCII string)
See `[SENSe:]CORRection:CONNection:CATalog?` on page 1380.

<CalKitName> Name of a cal kit (ASCII string) that is available for connector type <ConnectorType>.
See `[SENSe:]CORRection:CKIT:CATalog? <ConnectorName>`.

Example:
```
MMEM:LOAD:CKIT 'C:\Users\Public\
Documents\Rohde-Schwarz\Vna\
Calibration\Kits\New Kit 1.calkit'
```
Load the previously created cal kit file `New Kit 1.calkit` from the default cal kit directory.
```
CORR:CKIT:SEL 'N 50 Ohm', 'New Kit 1'
```
Assign the imported kit to the N 50 Ω connector type, assuming that the cal kit name stored in `New Kit 1.calkit` reads `New Kit 1`.

Manual operation: See "Cal Kit" on page 586

[SENSe:]CORRection:CKIT:<ConnType>:SELect <KitName>

Sets/gets the calibration kit to be used for connector type `<ConnType>`.

Tip: This command selects a cal kit by its name <CalKitName> – which can be ambiguous. Use `[SENSe:]CORRection:CKIT:<ConnType>:LSELect` to select a cal kit by its name *and* label.

Suffix:

<ConnType> N50 | N75 | PC35 | PC7 | PC292 | PC24 | PC 185 | SMA | USER1 | USER2
Connector type, see Table 7-16.
The R&S ZVR-compatible values `USER1 | USER2` refer to the user-defined connector types `UserConn<i>`, <i>=1,2. Define these connector types using `[SENSe:]CORRection:CONNection "UserConn<i>",...` before referring to them.
For other connector types (pre- or user-defined), you have to use one of the commands `[SENSe:]CORRection:CKIT:SELect` or `[SENSe:]CORRection:CKIT:LSELect`.

Parameters:

<KitName> Name of an existing calibration kit with connector type <ConnType>.
Use `[SENSe:]CORRection:CKIT:CATalog? <connector name>` or `[SENSe:]CORRection:CKIT:LCATalog? <connector name>` to query for these cal kits.

Example:

```
MMEM:LOAD:CKIT 'C:\Users\Public\
Documents\Rohde-Schwarz\Vna\
Calibration\Kits\New Kit 1.calkit'
```
Load the previously created cal kit file `New Kit 1.calkit` from the default cal kit directory.
`CORR:CKIT:N50:SEL 'New Kit 1'`
Assign the imported kit to the N 50 Ω connector type, assuming that the cal kit name stored in `New Kit 1.calkit` reads `New Kit 1`.

Manual operation: See "Available Cal Kits" on page 638

[SENSe:]CORRection:CKIT:STANdard:CATalog? <CalKitName>

Returns a list of all standards in calibration kit <CalKitName>.

Query parameters:

<CalKitName> Name of the cal kit (ASCII string).
Use `[SENSe:]CORRection:CKIT:CATalog?` to query for cal kit names.

Example: See `[SENSe:]CORRection:CKIT:CATalog?` on page 1319.

Usage: Query only

[SENSe:]CORRection:CKIT:STANdard:DATA? <ConnectorName>, <KitName>, <KitLabel>, <StandardType>, <DelayMode>[, <Port1>[, <Port2>]]

Returns the data of the related calibration standard (identified by <ConnectorName>, <KitName>, <KitLabel> and <StandardType>) and – optionally – restricted to the given port or ports.

The delay parameter is returned according to the specified <DelayMode>; see Table 7-14.

Query parameters:

<ConnectorName> Name of the connector type.
Use `[SENSe:]CORRection:CONNection:CATalog?` to query connector names.

<KitName>, <KitLabel> Name and label of the calibration kit.
Use `[SENSe:]CORRection:CKIT:LCATalog? <ConnectorName>` to query for these cal kits.

<StandardType> MMTHrough | MFTHrough | FFTHrough | MMLine | MMLine1 | MMLine2 | MMLine3 | MMLine4 | MMLine5 | MMLine6 | MFLine | MFLine1 | MFLine2 | MFLine3 | MFLine4 | MFLine5 | MFLine6 | FFLine | FFLine1 | FFLine2 | FFLine3 | FFLine4 | FFLine5 | FFLine6 | MMATten | MFATten | FFATten | MMSNetwork | MFSNetwork | FFSNetwork | MOPen | FOPen | MSHort | FSHort | MOSHort | MOSHort1 | MOSHort2 | MOSHort3 | FOSHort | FOSHort1 | FOSHort2 | FOSHort3 | MREFlect | FREFlect | MMTCh | FMTCh | MSMatch | FSMatch

Standard type; see Table 7-15.

<DelayMode> DELay | ELENgth

ELENgth (default) – ZVR compatible modeling
DELay – Keysight modeling

<Port1>[, <Port2>] Optional port restriction: one port number for one-port standards, two port numbers for two-port standards

Usage: Query only

[SENSe:]CORRection:CKIT:THRough:SELect [<CalKitName>]

Selects the cal kit or cal unit to be used for (unknown) Through measurements.

Parameters:

<CalKitName> For calibrations requiring characterized Throughs, specify the name of a suitable mechanical cal kit.
For calibrations requiring unknown Throughs, either specify the name of a calibration unit or 'Unknown' if you want to use a mechanical (reciprocal) 2-port standard or coupler.

7.3.14.5 [SENSe:]CORRection:CKIT... with labels

The following `[SENSe:]CORRection:CKIT...` commands identify the calibration kit to be used by a combination of its <CalkitName> and <CalkitLabel>. Typically, the serial number of the calibration kit serves as a calibration kit label. Due to their different labels, the analyzer can handle several calibration kits with identical names.

[SENSe:]CORRection:CKIT:LCATalog? 1325
[SENSe:]CORRection:CKIT:LDELete 1325
[SENSe:]CORRection:CKIT:LLABel 1325
[SENSe:]CORRection:CKIT:LSELect 1326
[SENSe:]CORRection:CKIT:<ConnType>:LSELect 1327
[SENSe:]CORRection:CKIT:STANdard:LCATalog? 1328
[SENSe:]CORRection:CKIT:<StandardType>:WLABel 1328
[SENSe:]CORRection:CKIT:<OnePortStandardType>:WLABel:SDATa? 1328
[SENSe:]CORRection:CKIT:<TwoPortStandard>:WLABel:SDATa? 1329

[SENSe:]CORRection:CKIT:LCATalog? [<ConnectorName>]

Queries all cal kits that are available for cal connector type <ConnectorName>, or for all cal connector types (if <ConnectorName> is omitted).

The return value is a string, containing a comma-separated list of cal kit names and labels (`'<kit1_name>,<kit1_label>,<kit2_name>,<kit2_label>, ...'`).

Query parameters:

<ConnectorName> Name of a connector type (ASCII string, optional).
Use `[SENSe:]CORRection:CONNection:CATalog?` to query connector names.

Example: See `[SENSe:]CORRection:CONNection:CATalog?`

Usage: Query only

[SENSe:]CORRection:CKIT:LDELete <CalKitName>, <KitLabel>

Deletes an imported or user-defined cal kit which is identified by its cal kit name and label.

Note: It is not possible to modify or store predefined or ideal kits.

Setting parameters:

<CalKitName> String parameter containing an imported or user-defined calibration kit available on the analyzer.

<KitLabel> String parameter containing the label of an imported or user-defined calibration kit available on the analyzer.

Example: See `[SENSe:]CORRection:CKIT:LLABel`

Usage: Setting only

[SENSe:]CORRection:CKIT:LLABel <CalKitName>, <KitLabel>[, <NewKitLabel>]

Assigns a calibration kit label to an imported or user-defined calibration kit or renames an existing calibration kit label.

Parameters:

<CalKitName> String parameter containing an imported or user-defined calibration kit available on the analyzer.

<KitLabel> String parameter containing the current calibration kit label.

<NewKitLabel> String parameter containing the new calibration kit label.

Example:

```
CORR:CKIT:FOP:WLAB 'N 50 Ohm','New Kit 1',
'Test kit created today', '',
0,4000000000,0,0,50,99,-2.3,0.22,0,0,0,0,0
```
Create a new cal kit "New Kit 1" labelled "Test kit created today" and assign an open (f) standard for the N 50 Ω connector type with specific properties.
```
CORR:CKIT:LLAB 'New Kit 1', 'Test kit created
today', '2012-05-25'
```
Change the label of the previously created kit.
```
CORR:CKIT:LLAB? 'New Kit 1', '2012-05-25'
```
Check the label.
```
MMEMory:STORe:CKIT:
WLABel 'New Kit 1', '2012-05-25', 'C:
\Users\Public\Documents
\Rohde-Schwarz\Vna\Calibration\Kits\New Kit 1
(2012-05-25).calkit'
```
Store the data for the labelled cal kit to the cal kit file
```
New Kit 1 (2012-05-25).calkit.
CORR:CKIT:LDEL 'New Kit 1', '2012-05-25'
```
Delete the kit. from the internal memory.
```
MMEMory:LOAD:CKIT 'C:\Users\Public\Documents
\Rohde-Schwarz\Vna\Calibration\Kits\New Kit 1
(2012-05-25).calkit'
```
Re-load the kit.

[SENSe:]CORRection:CKIT:LSELect <ConnectorName>, <KitName>, <KitLabel>

Sets or queries the calibration kit to be used for connector type <ConnectorName>.

Selects a calibration kit by its connector type name, name, and label.

Parameters:

<ConnectorName> Connector type name (ASCII string)
See [SENSe:]CORRection:CONNection:CATalog? on page 1380.

<KitName>, <KitLabel> Name and label of an existing cal kit for connector type <ConnectorName>.
See [SENSe:]CORRection:CKIT:LCATalog? <ConnectorName>.

Example:
```
MMEM:LOAD:CKIT 'C:\Users\Public\
Documents\Rohde-Schwarz\Vna\
Calibration\Kits\New Kit 1 (123456).calkit'
```
Load the previously created cal kit file `New Kit 1 (123456).calkit` from the default cal kit directory.
```
CORR:CKIT:LSEL 'N 50 Ohm', 'New Kit 1',
'123456'
```
Assign the imported kit to the N 50 Ω connector type, assuming that the cal kit name stored in `New Kit 1 (123456).calkit` reads `New Kit 1` and that its label reads `123456`.

[SENSe:]CORRection:CKIT:<ConnType>:LSELect <KitName>, <KitLabel>

Sets/gets the calibration kit to be used for connector type `<ConnType>`.

The kit is identified by its name and label.

Suffix:

<ConnType> N50 | N75 | PC35 | PC7 | PC292 | PC24 | PC 185 | SMA | USER1 | USER2
Connector type, see Table 7-16.
The R&S ZVR-compatible values `USER1 | USER2` refer to the user-defined connector types `UserConn<i>`, <i>=1,2. Define these connector types using `[SENSe:]CORRection:CONNection "UserConn<i>",...` before referring to them.
For other connector types (pre- or user-defined), use `[SENSe:]CORRection:CKIT:LSELect`.

Parameters:

<KitName>, <KitLabel> Name and label of an existing calibration kit with connector type <ConnType>.
Use `[SENSe:]CORRection:CKIT:LCATalog? <connector name>` to query for these cal kits.

Example:
```
MMEM:LOAD:CKIT 'C:\Users\Public
\Documents\Rohde-Schwarz
\Vna\Calibration\Kits\New Kit 1
(123456).calkit'
```
Load the previously created cal kit file `New Kit 1 (123456).calkit` from the default cal kit directory.
```
CORR:CKIT:N50:LSEL 'New Kit 1', '123456'
```
Assign the imported kit to the N 50 Ω connector type, assuming that the cal kit name stored in `New Kit 1 (123456).calkit` reads `New Kit 1` and that its label reads `123456`.

Manual operation: See "Available Cal Kits" on page 638

[SENSe:]CORRection:CKIT:STANdard:LCATalog? <KitName>, <KitLabel>

Returns a list of all standards in a given calibration kit.

Query parameters:

<KitName>, <KitLabel>
Cal kit name and label (ASCII strings)
Use `[SENSe:]CORRection:CKIT:LCATalog?` to query cal kit names and labels.

Example: See `[SENSe:]CORRection:CONNection:CATalog?`

Usage: Query only

[SENSe:]CORRection:CKIT:<StandardType>:WLABel <ConnType>, <CalKitName>, <CalKitLabel>, <StandardLabel>, <MinFreq>, <MaxFreq>, <DelayParam>, <Loss>, <Z0>, <C0>, <C1>, <C2>, <C3>, <L0>, <L1>, <L2>, <L3>, OPEN | SHORt | MATCh | <Resistance>[, <Port1>[, <Port2>]]
[SENSe:]CORRection:CKIT:<StandardType>:WLABel? <ConnType>[, <CalKitName>[, <CalKitLabel>[, <Port1>[, <Port2>]]]]

Defines the parameters of a non-ideal 1 port or 2-port calibration standard `<StandardType>`, where a particular cal kit can be addressed by name **and label**.

Apart from the additional `<CalKitLabel>` parameter, the syntax and semantics of this command is identical to `[SENSe:]CORRection:CKIT:<StandardType>`.

Suffix:

<StandardType>
<string>
Standard type
For a complete list of standards, refer to Table 7-15.

Example:
```
CORR:CKIT:FOP:WLAB 'N 50 Ohm','New Kit 1','123456', '',
0,4000000000,0,0,50,99,-2.3,0.22,0,0,0,0,0
```
Define the properties of the Open (f) standard for a N 50 Ω connector type in the calibration kit "New Kit 1" labelled "123456".
```
CORR:CKIT:FOP:WLAB? 'N 50 Ohm', 'New Kit 1', '123456'
```
Query the properties of the Open (f) standard for a N 50 Ω connector type in the calibration kit.

Manual operation: See "Add / Copy / Delete / View / Modify" on page 641

[SENSe:]CORRection:CKIT:<OnePortStandardType>:WLABel:SDATa? <ConnectorType>, <CalKitName>[, <CalKitLabel>[, <PhysPort>]]

Reads the S-parameter data for a particular **1-port** cal kit standard previously loaded from Touchstone file using `MMEMory:LOAD:CKIT:SDATa` or `MMEMory:LOAD:CKIT:SDATa:WLABel`.

The cal kit is identified by its name and label.

Query parameters:

<OnePort StandardType>	MOPen | FOPen | MSHort | FSHort | MOSHort | MOSHort1 | MOSHort2 | MOSHort3 | FOSHort | FOSHort1 | FOSHort2 | FOSHort3 | MREFlect | FREFlect | MMTCh | FMTCh | MSMatch | FSMatch Standard type. For more information see Table 7-15.
<ConnectorType>, <CalKitName>, <CalKitLabel>	Together with <StandardType> these parameters fully identify the related standard (see Parameter list.
<PhysPort>	Number of the physical port for which the S-parameter data is valid. Can be omitted if the data are valid for all ports.
Usage:	Query only

[SENSe:]CORRection:CKIT:<TwoPortStandard>:WLABel:SDATa?
<ConnectorType>, <CalKitName>[, <CalKitLabel>[, <SParameter>, <PhysPort1>, <PhysPort2>]]

Reads the S-parameter data for a particular **2-port** cal kit standard previously loaded from Touchstone file using `MMEMory:LOAD:CKIT:SDATa` or `MMEMory:LOAD:CKIT:SDATa:WLABel`.

The cal kit is identified by its name and label.

Query parameters:

<TwoPortStandard>	MMTHrough | MFTHrough | FFTHrough | MMLine | MMLine1 | MMLine2 | MMLine3 | MFLine | MFLine1 | MFLine2 | MFLine3 | FFLine | FFLine1 | FFLine2 | FFLine3 | MMATten | MFATten | FFATten | MMSNetwork | MFSNetwork | FFSNetwork Standard type. For more information see Table 7-15.
<ConnectorType>, <CalKitName>, <CalKitLabel>	Together with <StandardType> these parameters fully identify the related standard (see Parameter list.
<SParameter>	S11 | S12 | S21 | S22 S-parameter of the 2-port standard.
<PhysPort1>, <PhysPort2>	Numbers of the physical ports for which the S-parameter data is valid. Can be omitted if the data are valid for all port pairs.
Usage:	Query only

7.3.14.6 [SENSe:]CORRection:COLLect...

The `[SENSe:]CORRection:COLLect...` commands control the system error correction and measurement receiver (power) calibration.

[SENSe<Ch>:]CORRection:COLLect[:ACQuire]:SELected ... 1331
[SENSe<Ch>:]CORRection:COLLect:AUTO ... 1333
[SENSe<Ch>:]CORRection:COLLect:AUTO:ASSignment<Asg>:ACQuire ... 1334
[SENSe<Ch>:]CORRection:COLLect:AUTO:ASSignment<Asg>:CHECk? ... 1334
[SENSe<Ch>:]CORRection:COLLect:AUTO:ASSignment:ALL:COUNt? ... 1335
[SENSe<Ch>:]CORRection:COLLect:AUTO:ASSignment:COUNt? ... 1336
[SENSe<Ch>:]CORRection:COLLect:AUTO:ASSignment<Asg>:DEFine ... 1336
[SENSe<Ch>:]CORRection:COLLect:AUTO:ASSignment:DEFine:DEFault ... 1339
[SENSe<Ch>:]CORRection:COLLect:AUTO:ASSignment:DEFine:OPTimized ... 1340
[SENSe<Ch>:]CORRection:COLLect:AUTO:ASSignment<Asg>:DEFine:TPORt ... 1341
[SENSe<Ch>:]CORRection:COLLect:AUTO:ASSignment:DEFine:TPORt:DEFault ... 1341
[SENSe<Ch>:]CORRection:COLLect:AUTO:ASSignment:DELete:ALL ... 1342
[SENSe:]CORRection:COLLect:AUTO:CKIT ... 1342
[SENSe:]CORRection:COLLect:AUTO:CKIT:PASSword ... 1343
[SENSe:]CORRection:COLLect:AUTO:CKIT:PORTs ... 1343
[SENSe:]CORRection:COLLect:AUTO:CKIT:PORTs:ADD ... 1344
[SENSe<Ch>:]CORRection:COLLect:AUTO:CONFigure ... 1345
[SENSe<Ch>:]CORRection:COLLect:AUTO:CPORt ... 1347
[SENSe:]CORRection:COLLect:AUTO:EXPLicit ... 1347
[SENSe:]CORRection:COLLect:AUTO:MCONnect ... 1348
[SENSe<Ch>:]CORRection:COLLect:AUTO:PORTs ... 1348
[SENSe:]CORRection:COLLect:AUTO:PORTs:CONNection? ... 1349
[SENSe<Ch>:]CORRection:COLLect:AUTO:PORTs:TYPE ... 1350
[SENSe<Ch>:]CORRection:COLLect:AUTO:POWer ... 1351
[SENSe<Ch>:]CORRection:COLLect:AUTO:POWer:DEFault ... 1351
[SENSe<Ch>:]CORRection:COLLect:AUTO:REPeat ... 1352
[SENSe<Ch>:]CORRection:COLLect:AUTO:SAVE ... 1352
[SENSe<Ch>:]CORRection:COLLect:AUTO:TEMPcomp ... 1353
[SENSe<Ch>:]CORRection:COLLect:AUTO:TERMination ... 1353
[SENSe<Ch>:]CORRection:COLLect:AUTO:TYPE ... 1353
[SENSe<Ch>:]CORRection:COLLect:AUTO:UTHRough ... 1354
[SENSe:]CORRection:COLLect:AVERage ... 1355
[SENSe:]CORRection:COLLect:CHANnels:ALL ... 1355
[SENSe:]CORRection:COLLect:CHANnels:MCTYpes ... 1355
[SENSe<Ch>:]CORRection:COLLect:CKIT:INSTall ... 1357
[SENSe<Ch>:]CORRection:COLLect:CKIT:LOAD ... 1357
[SENSe<Ch>:]CORRection:COLLect:CKIT:PORT<PhyPt>? ... 1357
[SENSe<Ch>:]CORRection:COLLect:CONNection<PhyPt> ... 1358
[SENSe<Ch>:]CORRection:COLLect:CONNection:GENDers ... 1359
[SENSe<Ch>:]CORRection:COLLect:CONNection:PORTs ... 1360
[SENSe<Ch>:]CORRection:COLLect:DELete ... 1360
[SENSe<Ch>:]CORRection:COLLect:DETector ... 1361
[SENSe<Ch>:]CORRection:COLLect:DISCard ... 1361
[SENSe<Ch>:]CORRection:COLLect:FIXTure[:ACQuire] ... 1362
[SENSe<Ch>:]CORRection:COLLect:FIXTure:EXPort ... 1363
[SENSe<Ch>:]CORRection:COLLect:FIXTure:IMPort ... 1363
[SENSe:]CORRection:COLLect:FIXTure:LMParameter[:STATe] ... 1364
[SENSe:]CORRection:COLLect:FIXTure:LMParameter:LOSS[:STATe] ... 1365
[SENSe<Ch>:]CORRection:COLLect:FIXTure:SAVE ... 1365
[SENSe<Ch>:]CORRection:COLLect:FIXTure:STARt ... 1365

[SENSe<Ch>:]CORRection:COLLect:LOAD:SELected 1366
[SENSe<Ch>:]CORRection:COLLect:METHod:DEFine 1367
[SENSe:]CORRection:COLLect:PMETer:ID 1369
[SENSe<Ch>:]CORRection:COLLect:SAVE:SELected:DEFault 1369
[SENSe<Ch>:]CORRection:COLLect:SAVE:SELected[:DUMMy] 1370
[SENSe<Ch>:]CORRection:COLLect:SCONnection<PhyPt> 1371
[SENSe:]CORRection:COLLect:SWEep:CONTinuous 1371

[SENSe<Ch>:]CORRection:COLLect[:ACQuire]:SELected <Type>, <TestPort>[, <SecondPortOrAdapter>[, <Dispersion>[, <DelayTimePhase>]]]

Starts the acquisition of measurement data for the selected standard and port(s). The standards are reflection or transmission standards and can be connected to arbitrary analyzer ports.

Note:

- The calibration measurement has a variable timeout: *Timeout = (Sweep time / Number of sweep points) * 3 + 0.1 s*
  The timeout may be reached, e.g., while the measurement waits for the specified trigger event. It is used for all standard measurements including power calibration sweeps.
- For a sliding match, the R&S ZNB can acquire measurement data for up to 20 positions per port. Multiple calls of `[SENSe<Ch>:]CORRection:COLLect[:ACQuire]:SELected SLIDe, <TestPort>` for the same test port, implicitly increases the position until position 20 has been recorded. Subsequent calls start over at position 1, overwriting the previously acquired data.

Suffix:

<Ch> Channel number of the calibrated channel

Setting parameters:

<Type> THRough | OPEN | SHORt | OSHort | OSHort1 | OSHort2 | OSHort3 | MATCh | NET | ATT | REFL | SLIDe | ISOLation | LINE | LINe1 | LINe2 | LINe3 | LINe4 | LINe5 | LINe6 | UTHRough | POWer | TERMination | EATTenuator

Standard types: Through, Open, Short, Match, Symmetric Network (NET), Attenuation (ATT), Reflect, Sliding Match (SLIDe), Line1 (LINE1 and LINE are synonymous), Line2 to Line6 (esp. for TRL calibration), Offset Short 1 to 3 (OSHort), Unknown Through, power meter (in a SMARTer Cal).
ISOLation is not a physical standard: to measure the isolation (supported for transmission normalization and TOSM only!), it is recommended to terminate the related test ports suitably (e.g. with 50 Ω loads).
TERMination represents a receiver noise measurement in channel <Ch>, with a matched load connected to receiving port <TestPort>. This is part of a noise figure calibration using mechanical calibration standards (with option R&S ZNB-K30).

EATTenuator represents an external attenuation (between source and DUT during measurement) in a noise figure channel <Ch>. This step is part of a noise figure calibration (with option R&S ZNB-K30).

<TestPort> Test port number.
For a transmission standard or an adapter that is used as a "through", the input and output port numbers must be specified.
For reflection standards, only one port number is required.

*RST: n/a

<SecondPortOrAdapter> For a transmission standard or an adapter that is used as a "through", this parameter specifies the second test port.
For reflection measurements with an adapter connected between port and standard, set it to ON.
Otherwise set it to OFF or simply omit it.

<Dispersion> Optional status parameter for UTHRough standard:

OFF (0)
Standard is non-dispersive

ON (1)
Standard is dispersive

*RST: OFF (0)

<DelayTimePhase> Delay time or phase for UTHRough standard (optional):

AUTO
The analyzer determines the delay time or phase during the calibration sweep
Automatic determination of the phase
The UOSM algorithm provides the transmission factor of the unknown through standard up to an ambiguous sign. This yields the two alternative phase values displayed in the calibration wizard; see Unknown Through Standard.
In remote control, the analyzer performs a plausibility check to determine the correct phase. No manual selection is necessary. The check starts at the first sweep point, using the transmission factor with negative phase. The analyzer measures the phase at the subsequent sweep points, assuming that the phase difference between any two consecutive points is less than 90 deg. From these phase values, the analyzer calculates a linear extrapolation and derives an estimate for the DC phase limit. If this DC phase is in the vicinity of ... -180 deg, +180 deg, ... then the transmission factor with negative phase is adopted. If the DC phase is in the vicinity of ... 0 deg, +360 deg, ... then the transmission factor with inverted sign (corresponding to a 180 deg phase shift) is adopted.

<delay or phase>
Delay time in ps (for non-dispersive standards) or phase at the start frequency of the sweep in deg (for dispersive standards). If an estimate of the start phase is entered, the analyzer uses the **calculated** value which is closest to the estimate.

*RST: AUTO

Example: See `[SENSe<Ch>:]CORRection:COLLect:SAVE:SELected[:DUMMy]` or Section 8.2.5.5, "Adapter removal", on page 1679.

Usage: Setting only

Manual operation: See "Start Cal Sweep" on page 589

[SENSe<Ch>:]CORRection:COLLect:AUTO <Characterization>, <TestPort1>[, <TestPort2>[...]]

Selects and initiates an automatic calibration for the specified test ports using a single, auto-detected port assignment.

Tip:

- If the test setup contains a high attenuation, the analyzer may fail to detect the cal unit ports connected to each of its ports. In this case use the extended command `[SENSe<Ch>:]CORRection:COLLect:AUTO:PORTs`.
- Use `[SENSe<Ch>:]CORRection:COLLect:AUTO:TYPE` if you want to specify a particular calibration type for the automatic calibration.
- If several calibration units are connected, use `SYSTem:COMMunicate:RDEVice:AKAL:ADDRess` to select a unit for the calibration.
- Use `[SENSe<Ch>:]CORRection:COLLect:AUTO:CONFigure` to prepare an automatic calibration with multiple port assignments.

Suffix:

<Ch> Channel number of the calibrated channel

Setting parameters:

<Characterization> Location of the characterization (cal kit file) to be used for the automatic calibration (string parameter):
– If an empty string (`' '`) is specified, the factory cal kit file stored in the active calibration unit is used.
– A VNA cal kit file name `*.calkit` without path refers to a specific cal kit file stored in the internal memory of the active calibration unit.
– A VNA cal kit file name without path but prefixed with "SD:" refers to a specific cal kit file stored on the SD card inserted at the active calibration unit.
– A VNA cal kit file name `*.calkit` with path refers to a specific cal kit file stored in an arbitrary directory on the analyzer.

<TestPort1>, <TestPort2>, ... Test port numbers. For an n-port automatic calibration, n arbitrary (not necessarily consecutive) port numbers must be specified. The analyzer automatically detects the calibration unit ports connected to each analyzer port.

Example: `CORR:COLL:AUTO '', 1, 2, 4`
Perform an automatic 3-port calibration at test ports 1, 2, and 4 using the calibration unit's default calibration kit file and arbitrary test ports of the cal unit.

Usage: Setting only

Manual operation: See "Apply/Cancel" on page 576

[SENSe<Ch>:]CORRection:COLLect:AUTO:ASSignment<Asg>:ACQuire

Starts the automatic calibration sweep for the indicated channel and port assignment.

A complete, valid set of port assignments must be defined before you can initiate a calibration. See Section 4.5.5.6, "Multiple port assignments", on page 185 for background information.

Suffix:
<Ch> Number of the calibrated channel

<Asg> Number of the port assignment

Example: See `[SENSe<Ch>:]CORRection:COLLect:AUTO:ASSignment<Asg>:DEFine`

Usage: Event

Manual operation: See "Start Cal Sweep / Abort Sweep" on page 575

[SENSe<Ch>:]CORRection:COLLect:AUTO:ASSignment<Asg>:CHECk?

Uses port detection to check whether test ports and calibration unit ports are cabled according to assignment <Asg>.

Note that since firmware V4.25, the created port assignments also depend on the global `[SENSe:]CORRection:COLLect:AUTO:MCONnect` setting.

- If set to ON (Use Reduced Number of Throughs for Cal Unit enabled), the firmware creates a minimum sequence of port assignments with common port `[SENSe<Ch>:]CORRection:COLLect:AUTO:CPORt` ("reduced Through port assignments").
- If set to OFF (default), the sequence of port assignments starts with the "reduced Through port assignments", followed by additional port assignments covering the remaining Through measurements.

Measuring the "reduced Through port assignments" is mandatory, measuring additional ones is optional.

The number of created assignments can be queried using the `[SENSe<Ch>:]CORRection:COLLect:AUTO:ASSignment:COUNt?` command. The assignments themselves can then be queried using `[SENSe<Ch>:]CORRection:COLLect:AUTO:ASSignment<Asg>:DEFine?`.

The query returns incorrect and undetected connections, see the example below.

Suffix:

<Ch> Channel number

<Asg> Assignment number

Example:

```
*RST
:SENSe1:CORRECTION:COLLECT:AUTO:ASSignment:DELete:ALL
:SENSe1:CORRECTION:COLLECT:AUTO:ASSignment1:DEFine 1,1,2,4,4,6,3,2
:SENSe1:CORRECTION:COLLECT:AUTO:ASSignment1:DEFine? %1,1,2,4,4,6,3,2
```

Suppose that the cabling is correct. Then

```
:SENSE1:CORRECTION:COLLECT:AUTO:ASSignment1:CHECk?
```

returns

```
INCORRECT: NONE ; UNDETECTED: NONE
```

Suppose test port 2 was connected to calu port 4, and test port 4 was connected to calu port 6. Then

```
:SENSE1:CORRECTION:COLLECT:AUTO:ASSignment1:CHECk?
```

returns

```
INCORRECT: NONE ; UNDETECTED: 1,1,3,2
```

Suppose test port 1 was connected to calu port 3, test port 2 was connected to calu port 4, test port 4 was connected to calu port 6, and test port 3 was connected to calu port 1. Then

```
:SENSE1:CORRECTION:COLLECT:AUTO:ASSignment1:CHECk?
```

returns

```
INCORRECT: 1,1,3,2 ; UNDETECTED: NONE
```

Suppose test port 4 was connected to calu port 5. Then

```
:SENSE1:CORRECTION:COLLECT:AUTO:ASSignment1:CHECk?
```

returns

```
INCORRECT: 4,5 ; UNDETECTED: 1,1,3,2,2,4
```

Usage: Query only

Manual operation: See "Check Ports & Start Cal / Check Port Assignment" on page 576

[SENSe<Ch>:]CORRection:COLLect:AUTO:ASSignment:ALL:COUNt?

Returns the total number of port assignments of *all* calibrations.

See Section 4.5.5.6, "Multiple port assignments", on page 185 for background information.

Note:

Use `[SENSe<Ch>:]CORRection:COLLect:AUTO:ASSignment:COUNt?` to get the number of port assignments of the "current" calibration.

Suffix:
<Ch> Channel number

Example: See Section 8.2.5.3, "MultiCal (with calibration unit)", on page 1675

Usage: Query only

[SENSe<Ch>:]CORRection:COLLect:AUTO:ASSignment:COUNt?

Returns the number of port assignments of the "current" calibration, i.e. the calibration last created using `[SENSe<Ch>:]CORRection:COLLect:AUTO:CONFigure`.

This is particulary useful if the default port assignments were established using `[SENSe<Ch>:]CORRection:COLLect:AUTO:ASSignment:DEFine:DEFault`. See Section 4.5.5.6, "Multiple port assignments", on page 185 for background information.

Note:

In MultiCal scenarios, use `[SENSe<Ch>:]CORRection:COLLect:AUTO:ASSignment:ALL:COUNt?` to get the number of port assignments of all calibrations.

Suffix:
<Ch> Channel number

Example: See `[SENSe<Ch>:]CORRection:COLLect:AUTO:ASSignment<Asg>:DEFine` on page 1336

Usage: Query only

Manual operation: See "Port Assignment (manual)" on page 572

[SENSe<Ch>:]CORRection:COLLect:AUTO:ASSignment<Asg>:DEFine
<TestPort1>, <CalUnitPort1>[, <TestPort2>]...

Manually defines port assignment no. <Asg> for channel no. <Ch>.

With manual configuration a non-minimal set of port assignments can be created, which may increase measurement accuracy. On the other hand, it is up to the user to ensure that the created set of port assignments is complete and valid for the chosen calibration type (see `[SENSe<Ch>:]CORRection:COLLect:AUTO:CONFigure`).

Note that during the corresponding calibration sweep the R&S ZNB expects the physical port connections to be established exactly as defined by the port pairs.

Use

- `[SENSe<Ch>:]CORRection:COLLect:AUTO:ASSignment<Asg>:DEFine:TPORt` command to take advantage of auto-detection of port connections.

- `[SENSe<Ch>:]CORRection:COLLect:AUTO:ASSignment:DEFine:DEFault` or `[SENSe<Ch>:]CORRection:COLLect:AUTO:ASSignment:DEFine:TPORt:DEFault` to create the default assignments instead.

See Section 4.5.5.6, "Multiple port assignments", on page 185 for background information.

Suffix:

<Ch>	Number of the channel
<Asg>	Number of the port assignment

Parameters:

<TestPort1>	First test port number
<CalUnitPort1>	Number of the calibration unit port that is assigned to the first test port
<TestPort2>	Second test port number ...

Example: Let's perform a full 3-port calibration with a two-port calibration unit, using factory characterization and the port assignments given in the table below.
Prerequisite: the adequate calibration unit was selected before using `SYSTem:COMMunicate:RDEVice:AKAL:ADDRess`
`SENSe1:CORRection:COLLect:AUTO:ASSignment:DELete:ALL`
Deletes all available port assignments.
`SENSe1:CORRection:COLLect:AUTO:CONFigure 'FNPort',''`
Sets the automatic calibration to "Full n-port" with factory characterization.
`SENSe1:CORRection:COLLect:AUTO:ASSignment1:DEFine 2,1,3,2`
Creates port assignment 1 explicitely (no auto-detection).
`SENSe1:CORRection:COLLect:AUTO:ASSignment2:DEFine:TPORt 3,4`
Creates port assignment 2 implicitely (auto-detection).
Before starting the calibration sweep for port assignment 1, ensure test port 2 is connected to cal unit port 1 and test port 3 to cal unit port 2
`SENSe1:CORRection:COLLect:AUTO:ASSignment1:ACQUire`
Perfoms the calibration sweep for port assignment 1
Before starting the calibration sweep for port assignment 2, ensure test ports 3 and 4 are connected to the cal unit (in any order)
`SENSe1:CORRection:COLLect:AUTO:ASSignment2:ACQUire`
Perfoms the calibration sweep for port assignment 2; auto-detects the existing port-connections at runtime
`SENSe1:CORRection:COLLect:AUTO:SAVE`
Checks whether the acquired calibration data are sufficient to calculate the system error correction. If yes, applies them to the selected channel.
See Section 8.2.5.3, "MultiCal (with calibration unit)", on page 1675 for a MultiCal example.

Manual operation: See "Port Assignment (manual)" on page 572

Test Port	Port Assignment 1	Port Assignment 2
2	Cal Unit Port 1	-
3	Cal Unit Port 2	auto-detected
4	-	auto-detected

[SENSe<Ch>:]CORRection:COLLect:AUTO:ASSignment:DEFine:DEFault <TestPort1>[, <TestPort2>]...

Creates the default port assignment(s) for the specified test ports.

Note that since firmware V4.25, the created port assignments also depend on the global `[SENSe:]CORRection:COLLect:AUTO:MCONnect` setting.

- If set to ON (Use Reduced Number of Throughs for Cal Unit enabled), the firmware creates a minimum sequence of port assignments with common port `[SENSe<Ch>:]CORRection:COLLect:AUTO:CPORt` ("reduced Through port assignments").
- If set to OFF (default), the sequence of port assignments starts with the "reduced Through port assignments", followed by additional port assignments covering the remaining Through measurements.

Measuring the "reduced Through port assignments" is mandatory, measuring additional ones is optional.

The number of created assignments can be queried using the `[SENSe<Ch>:]CORRection:COLLect:AUTO:ASSignment:COUNt?` command. The assignments themselves can then be queried using `[SENSe<Ch>:]CORRection:COLLect:AUTO:ASSignment<Asg>:DEFine?`.

Note that during the following calibration steps, the R&S ZNB expects the physical port connections to be established exactly as specified by the created port assignments. Use `[SENSe<Ch>:]CORRection:COLLect:AUTO:ASSignment<Asg>:CHECk?` to check the connections using port detection.

Use

- `[SENSe<Ch>:]CORRection:COLLect:AUTO:ASSignment:DEFine:TPORt:DEFault` on page 1341 to use auto-detection instead of defining "test port --> cal unit port"-connections explicitely
- `[SENSe<Ch>:]CORRection:COLLect:AUTO:ASSignment<Asg>:DEFine` or `[SENSe<Ch>:]CORRection:COLLect:AUTO:ASSignment<Asg>:DEFine:TPORt` to define (additional) port assignments manually

See Section 4.5.5.6, "Multiple port assignments", on page 185 for background information.

Suffix:
<Ch> Channel number

Setting parameters:
<TestPort1>, <TestPort2>, ... Test port numbers

Example: See `[SENSe<Ch>:]CORRection:COLLect:AUTO:ASSignment<Asg>:DEFine` on page 1336

Usage: Setting only

Manual operation: See "Set to Default Port Assignment" on page 572

[SENSe<Ch>:]CORRection:COLLect:AUTO:ASSignment:DEFine:OPTimized <TestPort1>[, <TestPort2>]...

Asks the analyzer to calculate optimized port assignments for automatic calibration of the specified test ports.

Optimization only makes a difference for matrix setups (see "Optimized port assignment" on page 280). For other setups, the optimized assignment is identical to the default assignment (see `[SENSe<Ch>:]CORRection:COLLect:AUTO:ASSignment:DEFine:DEFault`).

Note that since firmware V4.25, the created port assignments also depend on the global `[SENSe:]CORRection:COLLect:AUTO:MCONnect` setting.

- If set to ON (Use Reduced Number of Throughs for Cal Unit enabled), the firmware creates a minimum sequence of port assignments with common port `[SENSe<Ch>:]CORRection:COLLect:AUTO:CPORt` ("reduced Through port assignments").
- If set to OFF (default), the sequence of port assignments starts with the "reduced Through port assignments", followed by additional port assignments covering the remaining Through measurements.

Measuring the "reduced Through port assignments" is mandatory, measuring additional ones is optional.

The number of created assignments can be queried using the `[SENSe<Ch>:]CORRection:COLLect:AUTO:ASSignment:COUNt?` command. The assignments themselves can then be queried using `[SENSe<Ch>:]CORRection:COLLect:AUTO:ASSignment<Asg>:DEFine?`.

In general, the optimized port assignments depend on the matrix structure and on the structure of the selected cal unit (see `SYSTem:COMMunicate:RDEVice:AKAL:ADDRess`).

Note that during the following calibration steps, the R&S ZNB expects the physical port connections to be established exactly as specified by the created port assignments. Use `[SENSe<Ch>:]CORRection:COLLect:AUTO:ASSignment<Asg>:CHECk?` to check the connections using port detection.

Suffix:
<Ch> Channel number

Setting parameters:
<TestPort1>, <TestPort2>, ... Test port numbers

Usage: Setting only

Manual operation: See "Set to Optimized Assignment" on page 572

[SENSe<Ch>:]CORRection:COLLect:AUTO:ASSignment<Asg>:DEFine:TPORt <TestPort1>[, <TestPort2>]...

Similar logic as `[SENSe<Ch>:]CORRection:COLLect:AUTO:ASSignment<Asg>:DEFine`, but initially defines the port assignment only by its underlying test port set; the connected calibration unit ports are **auto-detected** at the start of the corresponding calibration sweep.

Suffix:
<Ch> Number of the channel

<Asg> Number of the port assignment

Parameters:
<TestPort1> First test port number

<TestPort2> Second test port number ...

Example: See `[SENSe<Ch>:]CORRection:COLLect:AUTO:ASSignment<Asg>:DEFine` on page 1336

Manual operation: See "Port Assignment (manual)" on page 572

[SENSe<Ch>:]CORRection:COLLect:AUTO:ASSignment:DEFine:TPORt:DEFault <TestPort1>[, <TestPort2>]...

Similar logic as `[SENSe<Ch>:]CORRection:COLLect:AUTO:ASSignment:DEFine:DEFault`, but initially defines the default port assignments only by their underlying test port set(s); the connected calibration unit ports are **auto-detected** at the start of the corresponding calibration sweep.

Note that since firmware V4.25, the defined port sets also depend on the global `[SENSe:]CORRection:COLLect:AUTO:MCONnect` setting. Their number can be queried using `[SENSe<Ch>:]CORRection:COLLect:AUTO:ASSignment:COUNt?`.

See Section 4.5.5.6, "Multiple port assignments", on page 185 for background information.

Suffix:
<Ch> Channel number

Setting parameters:
<TestPort1>, <TestPort2>, ... Test port numbers

Example: See `[SENSe<Ch>:]CORRection:COLLect:AUTO:ASSignment<Asg>:DEFine` on page 1336

Usage: Setting only

Manual operation: See "Set to Default Port Assignment" on page 572

[SENSe<Ch>:]CORRection:COLLect:AUTO:ASSignment:DELete:ALL

Deletes all port assignments (created using `[SENSe<Ch>:]CORRection:COLLect:AUTO:ASSignment<Asg>:DEFine`) and calibration definitions (created using `[SENSe<Ch>:]CORRection:COLLect:AUTO:CONFigure`) of an ongoing automatic calibration in channel <Ch>.

Suffix:
<Ch> Channel number of the calibrated channel

Example: See `[SENSe<Ch>:]CORRection:COLLect:AUTO:ASSignment<Asg>:DEFine`

Usage: Event

[SENSe:]CORRection:COLLect:AUTO:CKIT <Characterization>

Generates a characterization (cal kit file) with the specified name containing the cal kit data of the active calibration unit (selected via `SYSTem:COMMunicate:RDEVice:AKAL:ADDRess`). The cal kit file can be saved to a directory on the analyzer, to the calibration unit's internal (flash) memory or to an SD card inserted at the calibration unit (if available).

Note that this command can only be executed if the number of cal unit ports is less or equal to the number of test ports. Furthermore the command assumes the "canonical" assignment of cal unit ports to test ports: cal unit port 1 assigned to test port 1, cal unit port 2 assigned to test port 2 etc. The new command `[SENSe:]CORRection:COLLect:AUTO:CKIT:PORTs` allows for flexible assignments.

Setting parameters:
<Characterization> Location of the created characterization / cal kit file (string parameter, extension *.calkit):
If a path is specified, the file is saved to the analyzer's hard disk. The default directory (`MMEMory:CDIRectory`) is **not** used.
If only the file name is specified, the file is saved to the calibration unit's internal (flash) memory. The factory calibration data on the unit is not overwritten.
If the file name is prefixed by "SD:", the file is saved to the SD card inserted at the calibration unit (if available/accessible).

Example: `CORR:COLL:AUTO:CKIT 'AutoCalChar'`
Generate a cal kit file `AutoCalChar.calkit` for the active calibration unit and store it in the calibration unit.
`SENSe:CORRection:COLLect:AUTO:CKIT 'SD:test1'`
Generate a cal kit file `test1.calkit` for the active calibration unit and store it in the calibration unit.

Usage: Setting only

Manual operation: See "File name / Comment (Optional)" on page 647

[SENSe:]CORRection:COLLect:AUTO:CKIT:PASSword <Password>

Enters a password to enable a single password-protected action in the automatic calibration or in the characterization wizard. If password protection has been activated manually in the "Characterize Cal Unit" dialog, the password is required for any single execution of one the following commands, provided that a cal kit (characterization) file different from the active characterization is specified:

`[SENSe]:CORRection:COLLect:AUTO`

`[SENSe]:CORRection:COLLect:AUTO:CKIT`

`[SENSe]:CORRection:COLLect:AUTO:PORTs`

`[SENSe]:CORRection:COLLect:AUTO:PORTs:TYPE`

`[SENSe]:CORRection:COLLect:AUTO:TYPE`

Tip: You have to send `[SENSe:]CORRection:COLLect:AUTO:CKIT:PASSword` repeatedly if your command script uses several password-protected commands. You do not need a password to perform automatic calibrations using the active cal unit characterization.

Setting parameters:

<Password> Password (string parameter), as defined in manual control.

Example:

```
SENSe:CORRection:COLLect:AUTO:CKIT:PASSword
'My_password'
```

Enter a password `My_password` (assuming that password protection has been activated manually).

```
SENSe:CORRection:COLLect:AUTO '', 1, 2
```

Perform an automatic 2-port calibration at test ports 1 and 2 using the calibration unit's default calibration kit file and automatic port assignment.

```
SENSe:CORRection:COLLect:AUTO:CKIT:PASSword
'My_password'
```

Re-enter the password.

```
SENSe:CORRection:COLLect:AUTO '', 1, 2
```

Repeat the calibration.

Usage: Setting only

Manual operation: See "Authentication" on page 644

[SENSe:]CORRection:COLLect:AUTO:CKIT:PORTs <Characterization>, <TestPort1>, <CalUnitPort1>[, <TestPort2>...]

Generates a characterization (cal kit file) with the specified name containing the cal kit data of the active calibration unit (`SYSTem:COMMunicate:RDEVice:AKAL:ADDRess`). The cal kit file can be saved to a directory on the analyzer, to the calibration unit's internal (flash) memory or to an SD card inserted at the calibration unit (if available).

Similar logic as [SENSe:]CORRection:COLLect:AUTO:CKIT, but with flexible port assignment.

Setting parameters:

<Characterization> Location of the created characterization / cal kit file (string parameter, extension *.calkit):
If a path is specified, the file is saved to the analyzer's hard disk. The default directory (MMEMory:CDIRectory) is **not** used.
If only the file name is specified, the file is saved to the calibration unit's internal (flash) memory. The factory calibration data on the unit is not overwritten.
If the file name is prefixed by "SD:", the file is saved to the SD card inserted at the calibration unit (if available/accessible).

<TestPort1> Number of first test port.

<CalUnitPort1> Number of the calibration unit port to whom the first test port (<TestPort1>) is assigned.

<TestPort2> Number of second test port...

Usage: Setting only

Manual operation: See "Test Port Assignment" on page 646

[SENSe:]CORRection:COLLect:AUTO:CKIT:PORTs:ADD <Characterization>, <TestPort1>, <CalUnitPort1>[, <TestPort2>]...

Extends or modifies an *existing* characterization of the active calibration unit (SYSTem:COMMunicate:RDEVice:AKAL:ADDRess).

This functionality is not available at the GUI.

Setting parameters:

<Characterization> Location of an existing characterization (e.g. created using [SENSe:]CORRection:COLLect:AUTO:CKIT:PORTs).
– A VNA cal kit file name *.calkit without path refers to a specific cal kit file stored in the internal memory of the active calibration unit.
– A VNA cal kit file name without path but prefixed with "SD:" refers to a specific cal kit file stored on the SD card inserted at the active calibration unit.
– A VNA cal kit file name *.calkit with path refers to a specific cal kit file stored in an arbitrary directory on the analyzer.

<TestPort1> Number of first test port.

<CalUnitPort1> Number of the calibration unit port, to whom test port <TestPort1> is assigned.

<TestPort2> Number of the second test port ...

Example:

Prerequisite:
`SYSTem:COMMunicate:RDEVice:AKAL:ADDRess 'MyCalU'`
Sets 'MyCalU' as the active calibration unit.
`SYSTem:COMMunicate:RDEVice:AKAL:PORTs? 'abc'`
Queries the ports of characterization 'abc'; returns something like '3,N 50 Ohm,MALE,4,N 50 Ohm,MALE'. In particular, calibration unit ports 1 and 2 are not yet characterized.
`SENSe:CORRection:COLLect:AUTO:CKIT:PORTs:ADD 'abc',1,2`
Extend characterization 'abc' with test port 1 assigned to port 2 of the cal unit.
`SYSTem:COMMunicate:RDEVice:AKAL:PORTs? 'abc'`
Now returns something like '2,N 50 Ohm,MALE,3,N 50 Ohm,MALE,4,N 50 Ohm,MALE'.
`SENSe:CORRection:COLLect:AUTO:CKIT:PORTs:ADD 'abc',1,2`
Repeat characterization of test port 2, e.g. because the connection cable wasn't fastened correctly ...

Usage: Setting only

[SENSe<Ch>:]CORRection:COLLect:AUTO:CONFigure <CalType>, <Characterization>[, <CalName>]

Selects a calibration type and a cal unit characterization (cal kit file) for an automatic calibration.

Suffix:

<Ch> Channel number of the calibrated channel

Parameters:

<CalType> FNPort | TOSM | PTOSm | FOPort | OPTPort | POPTport | FRTRans | FTRans | RTRans | REFL | PREFl | RSHort | PFOPort | PFNPort

Calibration Type

FNPort
Full n-port (UOSM) calibration

TOSM
Full n-port calibration with characterized Through.
If the selected cal unit characterization does not contain a Through characterization, the command silently falls back to a FNPort (UOSM) calibration.

PTOSM
TOSM with additional receiver power calibration

FOPort
Full one-port ("Refl OSM") calibration

OPTPort
"One Path Two Ports" calibration
If the selected cal unit characterization does not contain a Through characterization, the command will fail.

POPTport
OPTPort with additional receiver power calibration

FRTRans
Transmission normalization, bidirectional
If the selected cal unit characterization does not contain a Through characterization, the command will fail.

FTRans | RTRans
Transmission normalization, forward | reverse
If the selected cal unit characterization does not contain a Through characterization, the command will fail.

REFL
Reflection normalization, Open

PREFl
REFL with additional receiver power calibration (SMARTerCal)

RSHort
Reflection normalization, Short

PFOPort
FOPort, in combination with a receiver power calibration at all ports

PFNPort
FNPort with additional receiver power calibration (SMARTerCal)

<Characterization> Location of the characterization (cal kit file) to be used for the automatic calibration (string parameter):
– The empty string (`''`) refers to the factory calibration of (and stored on) the active calibration unit.
– A VNA cal kit file name `*.calkit` without path refers to a specific cal kit file stored in the internal memory of the active calibration unit.
– A VNA cal kit file name without path but prefixed with "SD:" refers to a specific cal kit file stored on the SD card inserted at the active calibration unit.
– A VNA cal kit file name `*.calkit` with path refers to a specific cal kit file stored in an arbitrary directory on the analyzer.
Note that for inline calibration systems this setting is ignored. The suitable, ICU-specific characterizations must be specified using `SYSTem:COMMunicate:RDEVice:AKAL:CKIT`.

Setting parameters:
<CalName>

Example: `SENSe1:CORRection:COLLect:AUTO:CONFigure FNPort,''`
Sets the automatic calibration to "Full n-port" with factory characterization.

Manual operation: See "Characterization" on page 568

[SENSe<Ch>:]CORRection:COLLect:AUTO:CPORt <PortNumber>

Defines the "Common Port" of the "reduced Through port assignments" created using `[SENSe<Ch>:]CORRection:COLLect:AUTO:ASSignment:DEFine:DEFault`, `[SENSe<Ch>:]CORRection:COLLect:AUTO:ASSignment:DEFine:TPORt:DEFault`, and similar commands.

If `[SENSe:]CORRection:COLLect:AUTO:MCONnect` is set to `ON`, the common port is the center of all Through measurements. For manually created port assignments, make sure that every port assignment contains it.

Suffix:
<Ch> Channel number

Setting parameters:
<PortNumber> Number of the common port

Usage: Setting only

Manual operation: See "Common Port" on page 569

[SENSe:]CORRection:COLLect:AUTO:EXPLicit <ExplicitSave>

Defines whether the cal unit part of an auto-calibration sequence (started using `[SENSe<Ch>:]CORRection:COLLect:AUTO:CONFigure`) must be terminated explicitly (using `[SENSe<Ch>:]CORRection:COLLect:AUTO:SAVE`).

Before firmware version 4.25, the cal unit part of an automatic calibration was always terminated when the first "non-calu" calibration command (e.g. `[SENSe<Ch>:]CORRection:COLLect:AUTO:POWer`) was executed. The "non-calu" command triggered an "auto-save" of the system error correction, a subsequent `[SENSe<Ch>:]CORRection:COLLect:AUTO:SAVE` causes an error. This is still the default behavior (`<ExplicitSave>` is `OFF`).

If `<ExplicitSave>` is set to `ON`, you can execute additional calu steps after executing a non-calu step.

Setting parameters:
<ExplicitSave>

OFF | 0
The first "non-calu" calibration command triggers an auto-save of the system error correction.
This is the default behavior.

ON | 1
The calu steps must be terminated by a `SENSe<Ch>:CORRection:COLLect:AUTO:SAVE`.
For backward compatibility reasons, the `SAVE` command automatically reverts `<ExplicitSave>` to `OFF` to restore the default behaviour.

*RST: OFF

Example: 4-port VNA, 2-port calu:

```
SENSe:CORRection:COLLect:AUTO:MCONnect OFF
SENSe:CORRection:COLLect:AUTO:EXPLicit ON
SENSe:CORRection:COLLect:PMETer:ID 1, 1
SENSe1:CORRection:COLLect:AUTO:CONFigure PFNP, ''
SENSe1:CORRection:COLLect:AUTO:ASSignment1:DEFine:DEFault 1, 2, 3, 4
SENSe1:CORRection:COLLect:AUTO:ASSignment1:ACQuire
SENSe1:CORRection:COLLect:AUTO:ASSignment2:ACQuire
SENSe1:CORRection:COLLect:AUTO:ASSignment3:ACQuire
SENSe1:CORRection:COLLect:AUTO:POWer 1
SENSe1:CORRection:COLLect:AUTO:ASSignment4:ACQuire
SENSe1:CORRection:COLLect:AUTO:ASSignment5:ACQuire
SENSe1:CORRection:COLLect:AUTO:ASSignment6:ACQuire
SENSe1:CORRection:COLLect:AUTO:SAVE
```

[SENSe:]CORRection:COLLect:AUTO:MCONnect <MinimumConnect>

Enables or disables the Reduced Through logic for every port assignment of an automatic calibration.

Note that for automatic Reduced Through calibrations to work, the port assignments must have a star topology, with a "Common Port" set using `[SENSe<Ch>:]CORRection:COLLect:AUTO:CPORt` on page 1347.

Setting parameters:

<MinimumConnect> ON (1), or OFF (0)

Manual operation: See "Use Reduced Number of Throughs for Cal Unit" on page 814

[SENSe<Ch>:]CORRection:COLLect:AUTO:PORTs <Characterization>, <TestPort1>, <CalUnitPort1>[, <TestPort2>]...

Selects and initiates an automatic calibration at arbitrary analyzer and calibration unit ports. A progress monitor for the calibration sweeps is displayed.

Tip:

- This command is necessary if the analyzer fails to detect the cal unit ports connected to each of its ports (e.g. because of a high attenuation in the test setup). If auto-detection works you can use the simpler command `[SENSe<Ch>:]CORRection:COLLect:AUTO`.
- If several calibration units are connected, use `SYSTem:COMMunicate:RDEVice:AKAL:ADDRess` to select a unit for the calibration.

Suffix:

<Ch> Channel number of the calibrated channel

Setting parameters:

<Characterization> Location of the characterization (cal kit file) to be used for the automatic calibration (string parameter):

– If an empty string (`' '`) is specified, the factory cal kit file stored in the active calibration unit is used.
– A VNA cal kit file name `*.calkit` without path refers to a specific cal kit file stored in the internal memory of the active calibration unit.
– A VNA cal kit file name without path but prefixed with "SD:" refers to a specific cal kit file stored on the SD card inserted at the active calibration unit.
– A VNA cal kit file name `*.calkit` with path refers to a specific cal kit file stored in an arbitrary directory on the analyzer.

<TestPort1> Test port numbers. For an n-port automatic calibration, n arbitrary (not necessarily consecutive) port numbers must be specified.

<CalUnitPort1> Port numbers of the cal unit. For an n-port automatic calibration, n arbitrary (not necessarily consecutive) port numbers must be specified. It is possible to combine arbitrary (not necessarily matching) pairs of analyzer and cal unit ports.

<TestPort2>

Example: `CORR:COLL:AUTO:PORT '', 1, 2, 2, 4, 4, 1`
Perform an automatic 3-port calibration at the analyzer ports 1, 2, and 4 using the calibration unit's default calibration kit file and ports 2, 4, and 1 of the cal unit.

Usage: Setting only

Manual operation: See "Apply/Cancel" on page 576

[SENSe:]CORRection:COLLect:AUTO:PORTs:CONNection?

Returns the assignment between the physical analyzer ports and the ports of the connected automatic calibration unit.

Example: `CORR:COLL:AUTO:PORT '', 1, 2, 2, 4, 4, 1`
Perform an automatic 3-port calibration at the analyzer ports 1, 2, and 4 using the calibration unit's default calibration kit file and ports 2, 4, and 1 of the cal unit.
`CORR:COLL:AUTO:PORT:CONN?`
Query the actual port assignment. If the cal unit is properly connected according to the previous command, the response is 1,2,2,4,3,0,4,1. A zero means that the corresponding analyzer port is not connected to any port of the calibration unit.

Usage: Query only

Manual operation: See "Detect Assignment" on page 572

[SENSe<Ch>:]CORRection:COLLect:AUTO:PORTs:TYPE <CalType>[, <Characterization>[, <TestPort1>[, <CalUnitPort1>[, <TestPort2>]...]]]

Selects and initiates an automatic calibration at arbitrary test and calibration unit ports. A progress monitor for the calibration sweeps is displayed.

Tip: This command is necessary if the analyzer fails to detect the cal unit ports connected to each of its ports (e.g. because of a high attenuation in the test setup). If autodetection works you can use the simpler command `[SENSe<Ch>:]CORRection:COLLect:AUTO:TYPE`.

Suffix:

<Ch> Channel number of the calibrated channel

Setting parameters:

<CalType> FNPort | TOSM | FOPort | OPTPort | FRTRans | FTRans | RTRans | REFL | RSHort | PFNPort

See `[SENSe<Ch>:]CORRection:COLLect:AUTO:CONFigure` on page 1345.

<Characterization> Location of the characterization (cal kit file) to be used for the automatic calibration (string parameter):
– If an empty string (`' '`) is specified, the factory cal kit file stored in the active calibration unit is used.
– A VNA cal kit file name `*.calkit` without path refers to a specific cal kit file stored in the internal memory of the active calibration unit.
– A VNA cal kit file name without path but prefixed with "SD:" refers to a specific cal kit file stored on the SD card inserted at the active calibration unit.
– A VNA cal kit file name `*.calkit` with path refers to a specific cal kit file stored in an arbitrary directory on the analyzer.

<TestPort1> Test port number. For an n-port automatic calibration, n arbitrary (not necessarily consecutive) port numbers must be specified. For a one path two port calibration (OPTPort), the first port no. denotes the source port (fully calibrated port).

<CalUnitPort1> Port numbers of the cal unit. For an n-port automatic calibration, n arbitrary (not necessarily consecutive) port numbers must be specified. It is possible to combine arbitrary (not necessarily matching) pairs of analyzer and cal unit ports.

<TestPort2> Second test port number

*RST: n/a

Example:

```
CORR:COLL:AUTO:PORT FNPort, '', 1, 2, 2, 4, 4, 1
```

Perform an automatic full 3-port calibration at test ports 1, 2, and 4 using the calibration unit's default calibration kit file and ports 2, 4, and 1 of the cal unit.

See also `[SENSe<Ch>:]CORRection:COLLect:AUTO:POWer`

Usage: Setting only

Manual operation: See "Calibration Type / Source" on page 569

[SENSe<Ch>:]CORRection:COLLect:AUTO:POWer <TestPort>

Initiates a receiver power calibration at physical port <TestPort>. This is the second step of an automatic SMARTerCal.

Note: The power calibration must be performed "after" the system error correction; see example below.

Suffix:
<Ch> Channel number of the calibrated channel

Setting parameters:
<TestPort> Test port number. The port number must match the port number selected via [SENSe:]CORRection:COLLect:PMETer:ID; see example.

Example: Suppose that a power meter no. 1 is configured and USB-connected to the analyzer.
SENSe:CORRection:COLLect:PMETer:ID 1, 2
Select power meter no. 1, to be connected to Port 2.
--> Connect the calibration unit between ports 1 and 2 of the analyzer,
SENSe:CORRection:COLLect:AUTO:PORTs:TYPE PFNPort, '', 1, 1, 2, 2
Perform an automatic full 2-port (UOSM) calibration at the analyzer test ports 1 and 2 using the calibration unit's default calibration kit file and default port assignments (ports 1 and 2 of the cal unit).
--> Remove the cal unit and connect the power meter to Port 2 of the analyzer.
SENSe:CORRection:COLLect:AUTO:POWer 2
Perform an additional receiver power calibration at Port 2. This completes the SMARTerCal.

Usage: Setting only

Manual operation: See "Start Cal Sweep / Abort Sweep" on page 610

[SENSe<Ch>:]CORRection:COLLect:AUTO:POWer:DEFault <TestPort>

Tells the automatic calibration logic to use default power meter data for a receiver power calibration at port <TestPort> instead of measuring with a power meter ([SENSe<Ch>:]CORRection:COLLect:AUTO:POWer on page 1351).

The power meter measurement can be done later ([SENSe<Ch>:]CORRection:COLLect:AUTO:REPeat on page 1352

Suffix:
<Ch> Channel number

Setting parameters:
<TestPort>

Example:

```
# define PUOSM Calibration
SENS1:CORR:COLL:PMET:ID 1, 1
SENSE1:CORRECTION:COLLECT:AUTO:ASSignment:DELete:ALL
SENSe1:CORRection:COLLect:AUTO:CONFigure PFNPort, ''
SENSE1:CORRECTION:COLLECT:AUTO:ASSignment1:DEFine 1,1, 2,2
# perform calibration with default power
SENSE1:CORRECTION:COLLECT:AUTO:ASSignment1:ACQUire
SENSe1:CORRection:COLLect:AUTO:POWer:DEF 1
# Repeat only power cal with powermeter
SENSe1:CORRection:COLLect:AUTO:CONFigure PFNPort, ''
SENSe1:CORRection:COLLect:AUTO:ASSignment1:DEFine 1, 1,2,2
SENSe1:CORRection:COLLect:AUTO:REPeat
SENSe1:CORRection:COLLect:AUTO:POWer 1
```

Usage: Setting only

[SENSe<Ch>:]CORRection:COLLect:AUTO:REPeat

Use this command to reuse measurement data of the active automatic calibration (if available) for the next automatic calibration. This data reuse allows an automatic calibration to be repeated without repeating all measurements.

Reusing existing calibration data is particularly useful for inline calibrations, whose OSM measurements can be repeated without reconnections, but whose unknown through measurements can not.

Note that this command does not work with "single-command calibrations" performed using `[SENSe<Ch>:]CORRection:COLLect:AUTO:PORTs:TYPE`.

Suffix:
<Ch> Channel number

Example: See Section 8.2.5.6, "Inline calibration", on page 1680

Usage: Event

[SENSe<Ch>:]CORRection:COLLect:AUTO:SAVE

Calculates the system error correction for automatic calibrations with Multiple port assignments, saves the data and applies the calibration to the active channel.

Requires successful calibration sweeps for all related port assignments (see `[SENSe<Ch>:]CORRection:COLLect:AUTO:ASSignment<Asg>:ACQuire`)

Suffix:
<Ch> Channel number of the calibrated channel

Example: See `[SENSe<Ch>:]CORRection:COLLect:AUTO:ASSignment<Asg>:DEFine`

Usage: Event

Manual operation: See "Apply/Cancel" on page 576

[SENSe<Ch>:]CORRection:COLLect:AUTO:TEMPcomp <Boolean>

Available if the calibration unit selected using `SYSTem:COMMunicate:RDEVice:AKAL:ADDRess` is an inline calibration system (ICS) only. Applies to all inline calibration units (ICUs) that are connected to the selected ICS and that are used with factory characterization data .

If set to ON (1), the R&S ZNB firmware adjusts the factory characterization data according to the ICU temperatures.

Suffix:
<Ch> Channel number

Parameters:
<Boolean>

Manual operation: See "Temperature Compensation" on page 569

[SENSe<Ch>:]CORRection:COLLect:AUTO:TERMination <ReceivingPort>

Acquires receiver noise data in Channel <Ch>, with a matched load connected to <ReceivingPort>. This is part of a noise figure calibration using a calibration unit.

Suffix:
<Ch> Channel number

Setting parameters:
<ReceivingPort> Port number of the receiving port

Example: `SENSe1:CORRection:COLLect:AUTO:TERMination 2`
Acquire noise data at receive port 2.

Usage: Setting only

Options: R&S ZNB-K30

[SENSe<Ch>:]CORRection:COLLect:AUTO:TYPE <CalType>[, <Characterization>[, <TestPort1>[, <TestPort2>]...]]

Selects and initiates an automatic calibration at arbitrary analyzer and cal unit ports. This command also selects the calibration type. A progress monitor for the calibration sweeps is displayed.

Tip: If the test setup contains a high attenuation the analyzer may fail to detect the cal unit ports connected to each of its ports. In this case use the extended command `[SENSe<Ch>:]CORRection:COLLect:AUTO:PORTs:TYPE`.

If several calibration units are connected, use `SYSTem:COMMunicate:RDEVice:AKAL:ADDRess` to select a unit for the calibration.

Suffix:
<Ch> Channel number of the calibrated channel

Setting parameters:

<CalType> FNPort | TOSM | FOPort | OPTPort | FRTRans | FTRans | RTRans | REFL | RSHort | PFOPort | POPTport | PFNPort

See `[SENSe<Ch>:]CORRection:COLLect:AUTO:CONFigure` on page 1345.

<Characterization> Location of the characterization (cal kit file) to be used for the automatic calibration (string parameter):
– The empty string (`' '`) refers to the factory calibration of (and stored on) the active calibration unit.
– A VNA cal kit file name `*.calkit` without path refers to a specific cal kit file stored in the internal memory of the active calibration unit.
– A VNA cal kit file name without path but prefixed with "SD:" refers to a specific cal kit file stored on the SD card inserted at the active calibration unit.
– A VNA cal kit file name `*.calkit` with path refers to a specific cal kit file stored in an arbitrary directory on the analyzer.

<TestPort1> Test port numbers. For an n-port automatic calibration, n arbitrary (not necessarily consecutive) port numbers must be specified.

<TestPort2> Second test port number of the analyzer

Example: `CORR:COLL:AUTO:TYPE FNPort, '', 1, 2, 4`
Perform an automatic full 3-port calibration at test ports 1, 2, and 4 using the calibration unit's default calibration kit file and arbitrary test ports of the cal unit.

Usage: Setting only

Manual operation: See "Calibration Type / Source" on page 569

[SENSe<Ch>:]CORRection:COLLect:AUTO:UTHRough <TestPort1>, <TestPort2>

With this command, you can perform the Unknown Through measurement during an automatic UOSM calibration.

Although it works for all calibration units, it is only useful if the calibration unit selected using `SYSTem:COMMunicate:RDEVice:AKAL:ADDRess` is an inline calibration system (ICS).

Suffix:

<Ch> Channel number

Setting parameters:

<TestPort1>, <TestPort2> VNA test port numbers

Example: See Section 8.2.5.6, "Inline calibration", on page 1680.

Usage: Setting only

[SENSe:]CORRection:COLLect:AVERage <Average>

Activates automatic averaging during calibration, which means that the VNA may perform multiple calibration sweeps and apply averaging to reduce trace noise. In contrast to manual averaging (see `[SENSe<Ch>:]AVERage[:STATe]`) the number of calibration sweeps is calculated automatically.

Parameters:
<Average> AUTO | MANual

Manual operation: See "Auto Averaging" on page 813

[SENSe:]CORRection:COLLect:CHANnels:ALL <Boolean>

Enables calibration of all channels in the active recall set. The command is valid for the following calibration methods:

- Manual system error correction
- Automatic system error correction (cal unit)
- Manual SMARterCal
- Automatic SMARTerCal (cal unit)

A scalar power calibration is not affected.

Parameters:
<Boolean> ON – calibrate all channels.
OFF – calibrate the active channel only.

Example: See `[SENSe<Ch>:]CORRection:COLLect:SAVE:SELected[:DUMMy]`

Manual operation: See "Calibrate all Channels" on page 584

[SENSe:]CORRection:COLLect:CHANnels:MCTYpes <Boolean>

Toggles the **M**ultiple **C**alibration **TYpes** mode that allows you to calibrate a subset of the available channels using channel-specific ports and calibration types (see Section 4.5.9, "Parallel calibration of multiple channels", on page 196).

Enable the MCTYpes mode before defining the calibrations to be performed using `[SENSe<Ch>:]CORRection:COLLect:METHod:DEFine`. Disable the MCTYpes mode after the calibrations were saved using `[SENSe<Ch>:]CORRection:COLLect:SAVE:SELected[:DUMMy]`. The latter must be called only *once*, with <Ch> referring to one of the channels to be calibrated (see the example below).

Parameters:
<Boolean>

Example: Suppose that there are 3 channels in the current recall set and you want to calibrate channels 1 and 2. For channel 1, you want to perform a Reflection Normalization (Short) on ports 1 and 2, for channel 2 an OSM calibration on port 2. This calibration requires measuring standards as shown in the table below. Proceed as follows:

```
:CORRection:COLLect:CHANnels:MCTYpes ON
```

Activate the MCTYPes mode.

```
:SENSe1:CORRection:COLLect:METHod:DEFine
'Parser Test SFK Ch1', RSHort, 1, 2
:SENSe2:CORRection:COLLect:METHod:DEFine
'Parser Test SFK Ch2', FOPort, 2
```

Declare the calibrations to be performed. Then connect the Short standard to port 1

```
:SENSe:CORRection:COLLect:ACQuire:SELected
SHORt, 1
```

Acquire calibration data for the short on port 1 (needed for calibration in channel 1). Then connect the Open standard to port 2.

```
:SENSe:CORRection:COLLect:ACQuire:SELected
OPEN, 2
```

Acquire calibration data for the open on port 2 (needed for calibration in channel 2). Then connect the Short standard to port 2.

```
:SENSe:CORRection:COLLect:ACQuire:SELected
SHORt, 2
```

Acquire calibration data for short on port 2 (needed for calibrations in channel 1 and 2). Then connect the Match standard to port 2.

```
:SENSe:CORRection:COLLect:ACQuire:SELected
MATCh, 2
```

Acquire calibration data for match on port 2 (needed for calibration in channel 2).

```
:SENSe1:CORRection:COLLect:SAVE:SELected
```

Complete all calibrations. This command is required only once and has to address one of the calibrated channels (channel 1 or 2 in this example).

```
:CORRection:COLLect:CHANnels:MCTYpes OFF
```

Deactivate the MCTYPes mode. Recommended for compatibility with manual operation.

	Port 1	**Port 2**
Channel 1 Reflection Normalization(Short) for ports 1 and 2	Short	Short
Channel 2 OSM for port 1	-	Open, Short, Match

[SENSe<Ch>:]CORRection:COLLect:CKIT:INSTall <CalKitFile>, <Gender>[, <Ports>]

Allows you to load cal kit data by gender:

- From the given file
- To the given ports or all ports

Use `MMEMory:CKIT:INFO?` on page 1259 to get information about cal kit files.

Suffix:

<Ch> Channel number

Setting parameters:

<CalKitFile> Path to the cal kit file, either absolute or relative to the current directory (see `MMEMory:CDIRectory`)

<Gender> MALE | FEMale | NGENder

<Ports> Either a comma-separated list of port numbers or ALL (optional)

Usage: Setting only

[SENSe<Ch>:]CORRection:COLLect:CKIT:LOAD <CalKitName>, <KitLabel>, <Gender>[, <Ports>]

Allows you to load cal kit data by name, label and gender:

- From the pool
- To the given ports or all ports

Suffix:

<Ch> Channel number

Setting parameters:

<CalKitName> The name of the cal kit to be loaded

<KitLabel> The label of the cal kit to be loaded

<Gender> MALE | FEMale | NGENder
The gender of the cal kit to be loaded

<Port> Either a comma-separated list of port numbers or ALL (optional)

Usage: Setting only

[SENSe<Ch>:]CORRection:COLLect:CKIT:PORT<PhyPt>? [<Detail>]

Queries the cal kit data assigned to the given port via

- `[SENSe<Ch>:]CORRection:COLLect:CKIT:LOAD`
  or `[SENSe<Ch>:]CORRection:COLLect:CKIT:INSTall`

Suffix:

<Ch> Channel number

<PhyPt> Port number

Query parameters:

<Detail> CONNector | LABel | NAME | GENDer
Queried property

Return values:

<Result>

Usage: Query only

[SENSe<Ch>:]CORRection:COLLect:CONNection<PhyPt> <ConTypeGender>

For manual calibrations of channel <Ch>, this command defines or queries the VNA-side connector type and gender at (the calibration plane of) physical port <PhyPt>.

Tip: Use `[SENSe<Ch>:]CORRection:COLLect:SCONnection<PhyPt>` to select a connector type by its name (ASCII string).

Suffix:

<Ch> Channel number of the calibrated channel

<PhyPt> Physical VNA port number

Parameters:

<ConTypeGender> N50Female | N50Male | N75Female | N75Male | PC35female | PC35male | PC7 | PC292female | PC292male | PC24female | PC24male | PC185female | PC185male | S7_16female | S7_16male | F75Female | F75Male | BNC50male | BNC50female | BNC75male | BNC75female | SMAFemale | SMAMale | CON431female | CON431male | UFEMale1 | UMALe1 | UFEMale2 | UMALe2

| <ConNameGender> (query only)
Enum values indicating the type and gender of the connectors (omitted for query). See column "Connector type+gender" in Table 7-16.
The R&S ZVR-compatible values `UFEMALE<i>` and `UMALE<i>` refer to the female and male variants of the user-defined connector type `USER<i>`, <i>=1,2. Define these connector types using `[SENSe:]CORRection:CONNection 'User Conn<i>',...` before referring to their gender variants.
Additional return values <ConNameGender> are created "on the fly", if connector and gender were specified using `SENSe<Ch>:CORRection:COLLect:SCONnection<PhyPt> <ConName>,<Gender>` and there is no corresponding enum value.
(Concatenation of <ConName> and <Gender> as string, blanks removed, all uppercase).
Note: If the analyzer is set to use the same connectors at all ports (`[SENSe<Ch>:]CORRection:COLLect:CONNection:PORTs ALL`), then setting the connector type applies to all ports. The gender of the connectors can be different though.

*RST: The connector type of the current instrument.

Example: `*RST; :CORR:COLL:CONN1 N75MALE; CONN4?`
Change the connector type at port 1 from N50FEMALE to N75MALE. The connector type at the other ports is also changed to N75, however, the gender (female) is maintained.
`CORR:COLL:CONN4?` returns `N75FEMALE`.

Manual operation: See "Connector" on page 302

Table 7-16: Connectors: String identifiers vs. enum values

Connector name* **(ASCII string)**	**Connector type <ConnType>****	**Connector type+gender (enum value)**
Predefined connector types:		
"N 50 Ohm"	N50	N50Female | N50Male
"N 75 Ohm"	N75	N75Female | N75Male
"3.5 mm"	PC35	PC35female | PC35male
"7 mm"	PC7	PC7
"2.92 mm"	PC292	PC292female | PC292male
"2.4 mm"	PC24	PC24female | PC24male
"1.85 mm"	PC185	PC185female | PC185male
"7-16"	n.a.	S7_16female | S7_16male
"Type F (75)"	n.a.	F75Female | F75Male
"BNC 50 Ohm"	n.a.	BNC50male | BNC50female
"BNC 75 Ohm"	n.a.	BNC75male | BNC75female
"SMA"	SMA	SMAFemale | SMAMale
"4.3-10"	n.a.	CON431female | CON431male
User-defined connector types:		
"UserConn1"	USER1	UFEMale1 | UMALe1
"UserConn2"	USER2	UFEMale2 | UMALe2
"<other>"	n.a.	<NAME+GENDER>

* At the VNA **GUI**, the string "Ohm" (case-insensitive) is represented by the symbol "Ω". Conversely, if you define a cal connector type containing a "Ω" at the GUI, the "Ω" is converted to the string "Ohm" at the remote interface.

** In commands `[SENSe:]CORRection:CKIT:<ConnType>:SELect` and `[SENSe:]CORRection:CKIT:<ConnType>:LSELect`

[SENSe<Ch>:]CORRection:COLLect:CONNection:GENDers <Gender>

Qualifies whether the genders at the test ports (but not their connector types) are equal or independent.

Suffix:
<Ch> Channel number of the calibrated channel

Parameters:

<Gender> ALL | SINGle

ALL – equal (uniform) genders. If the gender at one port is changed, the connector type at all other ports is changed accordingly.
SINGle – independent (possibly non-uniform) genders at the ports.

*RST: SINGle

Example: See `[SENSe<Ch>:]CORRection:COLLect:CONNection:PORTs`

Manual operation: See "Same Connector All Ports / Same Gender All Ports" on page 587

[SENSe<Ch>:]CORRection:COLLect:CONNection:PORTs <ConnectorType>

Qualifies whether the connector types at the test ports (but not their gender) are equal or independent. Some calibration types require uniform port connector types.

Suffix:

<Ch> Channel number of the calibrated channel

Parameters:

<ConnectorType> ALL | SINGle

ALL – equal (uniform) connector types. If the connector type at one port is changed, the connector type at all other ports is changed accordingly.
SINGle – independent (possibly non-uniform) connector types at the ports.

*RST: ALL

Example:

```
CORR:COLL:CONN:PORTS SING
```

Select independent connector types at the ports.

```
CORR:COLL:CONN1 N50MALE; CONN4 N75FEMALE;
CONN2?
```

Select independent connector types at ports 1 and 4. The connector type at port 2 is not changed; the query returns `N50FEMALE`.

Manual operation: See "Same Connector All Ports / Same Gender All Ports" on page 587

[SENSe<Ch>:]CORRection:COLLect:DELete [<CalName>]

Deletes system error correction data generated and stored previously.

Suffix:

<Ch> Channel number of the calibrated channel

Setting parameters:

<CalName> Name of the calibration (string parameter) defined together with the calibration type ([SENSe<Ch>:]CORRection:COLLect:METHod:DEFine).
ALL - the analyzer deletes all calibrations.
If nothing is specified the analyzer deletes the last system error correction stored by means of [SENSe<Ch>:]CORRection:COLLect:SAVE:SELected[:DUMMy].

*RST: -

Example:

```
CORR:COLL:METH:DEF 'Test',RSHort,1
```
Select a one-port normalization at port 1 with a short standard as calibration type.
```
CORR:COLL:SEL SHOR,1
```
Measure a short standard connected to port 1 and store the measurement results of this standard.
```
CORR:COLL:SAVE:SEL
CORR:COLL:DEL ALL
```
Calculate the system error correction data and apply it to the active channel, then delete the data.

Usage: Setting only

Manual operation: See "Apply" on page 591

[SENSe<Ch>:]CORRection:COLLect:DETector <CalName>[, <CalDetectorType>]

Selects the detector type during manual calibration <CalName>.

Suffix:

<Ch> Channel number

Parameters:

<CalName> Name of a manual calibration previously created e.g. using [SENSe<Ch>:]CORRection:COLLect:METHod:DEFine

<CalDetectorType> NORMal | AVERage

AVERage
Average detector, recommended for channels measuring traces using detectors other than the NORMal one
Automatically selected for noise figure calibrations.

NORMal
Normal (sample) detector, recommended for all other applications

*RST: NORMal

[SENSe<Ch>:]CORRection:COLLect:DISCard

Terminates a system error correction, discarding the acquired data.

Suffix:
<Ch> Channel number of the calibrated channel

Example:
```
:SENSe1:CORRection:COLLect:METHod:DEFine 'Test', TOSM, 1, 2
:SENSe1:CORRection:COLLect:ACQuire:SELected  THRough, 1, 2
:SENSe1:CORRection:COLLect:ACQuire:SELected  OPEN,    1
:SENSe1:CORRection:COLLect:ACQuire:SELected  SHORt,   1
:SENSe1:CORRection:COLLect:ACQuire:SELected  MATCh,   1
:SENSe1:CORRection:COLLect:ACQuire:SELected  OPEN,    2
:SENSe1:CORRection:COLLect:ACQuire:SELected  MATCh,   2
:SENSe1:CORRection:COLLect:DISCard
```

Usage: Event

[SENSe<Ch>:]CORRection:COLLect:FIXTure[:ACQuire] <StandardType>, <TestPort1>[, <TestPort2>]...

Starts a fixture compensation sweep in order to acquire measurement data for a test fixture that has its inner conductor terminated with the selected standards.

Suffix:
<Ch> Channel number of the corrected channel

Setting parameters:
<StandardType> OPEN | SHORt

Terminating standard type: open or short.

<TestPort1>

<TestPort2> Test port numbers. For a fixture compensation, n arbitrary (not necessarily consecutive) port numbers must be specified.

*RST: -

Example:

```
*RST; CORR:COLL:FIXT:LMP:LOSS OFF
```
Configure a fixture compensation measurement (for all channels): The analyzer performs an Auto Length (no loss) calculation.
```
CORR:COLL:FIXT:ACQ OPEN, 2; :CORR:COLL:FIXT:ACQ SHOR, 4
```
Perform a fixture compensation sweep at port 2, terminated with an open standard, and at port 4, terminated with a short.
```
CORR:COLL:FIXT:SAVE
```
Save and apply the compensation data.
```
CORR:COLL:FIXT:STAR
```
Prepare a new fixture compensation measurement, deleting the previous data for channel 1.
```
CORR:COLL:FIXT:LMP OFF
```
Select a Direct Compensation measurement (for all channels and traces).
```
CORR:COLL:FIXT:ACQ SHOR, 1, 3
```
Perform a fixture compensation sweep at ports 1 and 3, terminated with a short standard.
```
CALC2:PAR:SDEF 'Trc2', 'S22'
```
Create channel no. 2 with a trace named Trc2.
```
SENS2:CORR:COLL:FIXT:STAR
```
Prepare a fixture compensation measurement for channel 2. The channel 1 data is not affected.
```
SENS2:CORR:COLL:FIXT:ACQ SHOR, 1, 3
```
Repeat the previous fixture compensation sweep for channel 2.
```
SENS1:CORR:COLL:FIXT:SAVE; :SENS2:CORR:COLL:FIXT:SAVE
```
Save and apply the compensation data for both channels.
```
SENS1:CORR:OFFS3:DFC?; :SENS2:CORR:OFFS3:DFC?
```
Query whether the analyzer uses Direct Compensation results at port 3. The response is `1;1` (true for both channels).

Usage: Setting only

Manual operation: See "Measure Fixture wizard" on page 742

[SENSe<Ch>:]CORRection:COLLect:FIXTure:EXPort <FixtureFile>, <StandardType>, <TestPort1>[, <TestPort2>]...
[SENSe<Ch>:]CORRection:COLLect:FIXTure:IMPort <FixtureFile>, <StandardType>, <TestPort1>[, <TestPort2>...]

Loads/saves "Direct Compensation" data from/to the specified file.

The `EXPort` command first acquires the required data, just as a `[SENSe<Ch>:]CORRection:COLLect:FIXTure[:ACQuire]` would do.

"Direct Compensation" data files are standard Trace files, containing reflection parameter traces for the related port(s) and standard.

Table 7-17: Direct Compensation data

#TestPorts	File Type
1	s1p
>=1	csv

For 1-port Touchstone files (*.s1p) only a single test port can be specified - otherwise an error is raised.

Suffix:
<Ch> Channel number

Parameters:
<FixtureFile> Path to the "Direct Compensation" data file, either absolute or relative to the current directory (set/queried using `MMEMory:CDIRectory`)

Setting parameters:
<StandardType> OPEN | SHORt

Terminating standard type

<TestPort1>[, <TestPort2>, ...] Test port numbers. For a fixture compensation, n different (but not necessarily consecutive) port numbers must be specified.

Example:
```
*RST; SENSe1:CORR:COLL:FIXT:LMP OFF
```

Activate "Direct Compensation".
```
SENSe1:CORR:COLL:FIXT:EXP 'Traces\p12_short.csv', SHOR, 1, 2
```

Acquire "Direct Compensation" data for ports 1 and 2 (whose inner connectors have to be terminated with a Short) and save the acquired data to file.
In a subsequent measurement session you can load and apply these data as follows:
```
*RST;
:SENSe1:CORR:COLL:FIXT:LMP OFF
:SENSe1:CORR:COLL:FIXT:IMP 'Traces\p12short.csv', SHOR, 1, 2
:SENSe1:CORR:COLL:FIXT:SAVE
```

Usage: Setting only

Manual operation: See "Measure Fixture wizard" on page 742

[SENSe:]CORRection:COLLect:FIXTure:LMParameter[:STATe] <Boolean>

Selects an Auto Length (and Loss) calculation or a Direct Compensation.

Parameters:

<Boolean> 1 - Auto Length (and Loss), depending on the last `[SENSe<Ch>:]CORRection:COLLect:FIXTure:LMParameter:LOSS[:STATe]` setting
0 - Direct Compensation
*RST: 1

Example: See `[SENSe<Ch>:]CORRection:COLLect:FIXTure[:ACQuire]`

Manual operation: See "Direct Compensation" on page 742

[SENSe:]CORRection:COLLect:FIXTure:LMParameter:LOSS[:STATe] <Boolean>

Selects an Auto Length or an Auto Length and Loss calculation.

Parameters:

<Boolean> 1 - Auto Length and Loss
0 - Auto Length, no loss
*RST: 1

Example: See `[SENSe<Ch>:]CORRection:COLLect:FIXTure[:ACQuire]`

Manual operation: See "Auto Length / Auto Length and Loss" on page 741

[SENSe<Ch>:]CORRection:COLLect:FIXTure:SAVE

Completes a fixture compensation, storing and applying the acquired data.

Suffix:

<Ch> Channel number of the corrected channel

Example: See `[SENSe<Ch>:]CORRection:COLLect:FIXTure[:ACQuire]`

Usage: Event

Manual operation: See "Measure Fixture wizard" on page 742

[SENSe<Ch>:]CORRection:COLLect:FIXTure:STARt

Prepares the analyzer for fixture compensation comprising a single or a series of fixture compensation sweeps (`[SENSe<Ch>:]CORRection:COLLect:FIXTure[:ACQuire]`). Previous compensation data is deleted.

Suffix:

<Ch> Channel number of the corrected channel

Example: See `[SENSe<Ch>:]CORRection:COLLect:FIXTure[:ACQuire]`

Usage: Event

Manual operation: See "Measure Fixture wizard" on page 742

[SENSe<Ch>:]CORRection:COLLect:LOAD:SELected <CalGroupFile>, <Standard>, <TestPort1>[, <SecondPortOrAdapter>]

Reloads a set of previously acquired calibration data for a particular standard from a file in the cal pool. The loaded data may be combined with new calibration measurement data (`[SENSe<Ch>:]CORRection:COLLect[:ACQuire]:SELected`) in order to simplify and speed up the new calibration procedure. The channel settings for loaded and new calibrations (e.g. the number of sweep points) must be identical.

Note:

- The analyzer performs a consistency check for the loaded data. If the loaded file is incompatible with the channel settings of channel `<Ch>`, or if it does not contain data for the specified standard and port(s), a command error message (–100, "Command error;...") is generated.
- For a sliding match, the R&S ZNB can acquire and load measurement data for up to 20 positions per port. Multiple calls of `[SENSe<Ch>:]CORRection:COLLect:LOAD:SELected <CalGroupFile>, SLIDe, <TestPort>` for the same cal group file and test port, implicitly increases the position until position 20 has been recorded. Subsequent calls will start over at position 1, overwriting the previously loaded data.

Suffix:

<Ch> Channel number of the calibrated channel

Parameters:

<CalGroupFile> String parameter to specify the name of the loaded cal group file.

<Standard> THRough | OPEN | SHORt | OSHort | OSHort1 | OSHort2 | OSHort3 | MATCh | NET | ATT | REFL | SLIDe | ISOLation | LINE | LINe1 | LINe2 | LINe3 | LINe4 | LINe5 | LINe6 | UTHRough | POWer

See `[SENSe<Ch>:]CORRection:COLLect[:ACQuire]:SELected` on page 1331.

<TestPort1> Test port number. For a transmission standard (through, line, attenuation, symmetric network) or an adapter used as a "through" the input and output port numbers must be specified. For reflection standards, only one port number is required.

Setting parameters:

<SecondPortOrAdapter> For a transmission standard or an adapter used as a "through" this parameter specifies the second test port.
For reflection measurements with an adapter connected between port and standard, set it to ON.
Otherwise set it to OFF or simply omit it.

Example: Suppose that the cal pool contains a file `'Calgroup3.cal'` with a valid through calibration for the active channel no. 1, which you want to include in a new TOSM calibration for ports 1 and 2.

```
MMEM:CDIR DEF; CDIR 'Calibration\Data'
```

Go to the cal group directory.

```
:SENSe1:CORRection:COLLect:METHod:DEFine 'New Cal', TOSM, 1, 2
```

Define a new TOSM calibration.

```
SENSe1:CORRection:COLLect:LOAD:SELected 'Cal Group 3.cal', THROUGH, 1, 2
```

Load the through data from the cal pool file into the new calibration.

```
SENSe1:CORRection:COLLect:ACQuire:SELected OPEN, 1
```

Proceed with the new calibration measurements.

Manual operation: See "Start Cal Sweep" on page 589

[SENSe<Ch>:]CORRection:COLLect:METHod:DEFine <CalName>, <CalType>, <TestPort1>[, <TestPort2>[, <TestPort3orAdapterCalKit>[, <TestPorts>]...]]

Defines the calibration to be performed for channel `<Ch>`.

If you want to calibrate multiple channels in parallel, use

- `[SENSe:]CORRection:COLLect:CHANnels:ALL` ON or
- `[SENSe:]CORRection:COLLect:CHANnels:MCTYpes` ON

before executing this command. For background information, see Section 4.5.9, "Parallel calibration of multiple channels", on page 196.

Calibration data is acquired using `[SENSe<Ch>:]CORRection:COLLect[:ACQuire]:SELected`.

Suffix:

<Ch> Channel number

Parameters:

<CalName> Name of the calibration (string parameter). The name serves as a reference to delete a particular set of system correction data (`[SENSe<Ch>:]CORRection:COLLect:DELete`).

<CalType> ARTosm | ARPTosm | REFL | RSHort | FOPort | PFOPort | FRTRans | FTRans | RTRans | UTRans | OPTPort | POPTport | TOSM | TOM | TSM | TRM | TRL | TNA | UOSM | PTOSm | PTOM | PTSM | PTRM | PTRL | PTNA | PUOSm | MTOSm | MUOSm | MOSM | MTRL | NTRL | PNTRl

Calibration types; see list below.

<TestPort1> First calibrated test port number. For an n-port calibration type, n port numbers must be specified. If more than n numbers are defined, the spare numbers (the last ones in the list) are ignored. Entering less than n numbers causes an error message. For a one path two port calibration (`OPTPort`), the first port no. denotes the node port (fully calibrated port).

<TestPort2> Second calibrated port number.

<TestPort3 or AdapterCalKit> Either the third calibrated port number, or – for Adapter Removal calibrations only – the file path (relative to the current directory) where the analyzer shall store the adapter characterization obtained during the calibration.
The latter is optional. If a file path is provided, the characterization file (in Touchstone s2p format) is generated when the correction terms are finally calculated (`[SENSe<Ch>:]CORRection:COLLect:SAVE:SELected[:DUMMy]`).

<TestPorts> More ports to be calibrated.

Example:
```
:SENSE1:CORRECTION:COLLECT:METHOD:DEFine 'Test', ARTosm, 1,2, 'Adapter.s2p'
```
Initiate an Adapter Removal calibration for ports 1 and 2 on channel 1 and store the adapter characterization in `<MMEMORY:CDIR?>\Adapter.s2p`.
For another example, see `[SENSe<Ch>:]CORRection:COLLect:SAVE:SELected[:DUMMy]`.

Manual operation: See "Type/Source" on page 584

The supported calibration types are listed below.

Parameter	Type (GUI)
ARTosm	Adapter Removal
REFL | RSHort	Refl Norm Open | Refl Norm Short
FOPort	Full One Port ("Refl OSM")
FRTRans | UTRans	n-port calibration types "Trans Norm Both" (forward and reverse) | "Trans Norm" (unidirectional), n > 1 For calibration type UTRans, the VNA performs star-shaped Through measurements, with <TestPort1> used as the source port. For calibration type FRTRans, it performs all possible Through measurements between the specified test ports.
FTRans | RTRans	Two-port calibration type "Trans Norm" in "Forward" | "Reverse" direction, where "Forward" ("Reverse") means that the port with the smaller (larger) port number is the source port.
OPTPort	One Path Two Ports
TOSM	TOSM
UOSM	UOSM
TRL | NTRL | TOM | TSM | TRM | TNA	TRL | NIST Multiline TRL | TOM | TSM | TRM | TNA

Parameter	Type (GUI)
ARPTosm | POPTport | PTOSm | PUOSm | PTRL | PNRLI | PTOM | PTSM | PTRM | PTNA	One of the previous n-port calibration types, in combination with a receiver power calibration at test port <TestPort1> (SMARTerCal)
PFOPort	FOPort, in combination with a receiver power calibration at all ports (SMARTerCal)
MTOSm | MUOSm | MOSM | MTRL	METAS Calibration of type OSM, UOSM, OSM, or TRL

[SENSe:]CORRection:COLLect:PMETer:ID <PowerMeter>, <TestPort>

Selects an external power meter for the SMARTerCal and assigns it to an analyzer port.

Note: The command cannot be used unless a power meter is connected via GPIB bus, USB or LAN interface and configured in the "External Power Meters" dialog.

Parameters:

<PowerMeter> Number of external power meter. The parameters UP, DOWN, MIN, MAX are not available for this command.

Range: 1 to number of configured external generators
*RST: The power meter selection is not changed by a reset of the analyzer.

<TestPort> Test port number. During power calibration, the power meter is connected to this port.

*RST: 1 to the number of test ports

Example: See `[SENSe<Ch>:]CORRection:COLLect:AUTO:POWer`

Manual operation: See "Power Meter" on page 605

[SENSe<Ch>:]CORRection:COLLect:SAVE:SELected:DEFault

Generates a set of default correction data for the selected ports and calibration type. The default data set corresponds to a test setup which does not introduce any systematic errors; none of the measurement results acquired previously (`[SENSe<Ch>:]CORRection:COLLect[:ACQuire]:SELected`) is considered.

The main purpose of the default correction data set is to provide a dummy calibration which you can partly replace with your own, external correction data. You may have acquired the external data in a previous session or even on another instrument. If you want to use the external correction data on the analyzer, simply generate the default data set corresponding to your port configuration and calibration type and overwrite the default data. For details refer to the program example below.

Another use case is simulating a calibrated instrument with R&S ZNXSIM.

Tip:

Since FW V.3.50 generating default calibration data also works for frequency-converting channels.

Suffix:
<Ch> Channel number of the calibrated channel.

Example:
`CORR:COLL:METH:DEF 'Test',RSHort,1`
Select a one-port normalization at port 1 with a short standard as calibration type.
`CORR:COLL:SAVE:SEL:DEF`
Calculate a dummy system error correction for the normalization at port 1. The dummy system error correction provides the reflection tracking error term 'REFLTRACK'.
`CORR:CDAT? 'REFLTRACK',1,0`
Query the dummy system error correction term. The response is a 1 (written as `1,0` for the real and imaginary part) for each sweep point (no attenuation and no phase shift between the analyzer and the calibration plane).
`INIT:CONT OFF; :INIT; *WAI`
Stop the sweep to ensure correct transfer of calibration data.
`CORR:CDAT 'REFLTRACK',1,0,<ASCII_data>`
Replace the dummy system error correction term with your own correction data, transferred in ASCII format.
`INIT:CONT ON`
Restart the sweep in continuous mode.

Usage: Event

Manual operation: See "Apply" on page 591

[SENSe<Ch>:]CORRection:COLLect:SAVE:SELected[:DUMMy]

Calculates the system error correction data from the acquired measurement results (`[SENSe<Ch>:]CORRection:COLLect[:ACQuire]:SELected`), stores it and applies it to the calibrated channel(s). To avoid incompatibilities, older system error correction data is deleted unless it has been transferred into a cal pool (`MMEMory:STORe:CORRection <Ch>, '<file_name>'`).

Suffix:
<Ch> One of the calibrated channels
If `[SENSe:]CORRection:COLLect:CHANnels:ALL` is ON this suffix can be omitted

Example:
`CORRection:COLLect:METHod:DEFine 'Test',RSHort,1`
Select a one-port normalization at port 1 with a short standard as calibration type.
`CORRection:COLLect:CHANnels:ALL ON`
Enable calibration in all channels of the active recall set.
`CORRection:COLLect:SELected SHOR,1`
Measure a short standard connected to port 1 and store the measurement results of this standard.
`CORRection:COLLect:SAVE:SELected`
Calculate the system error correction data and apply it to all channels of the active recall set.

Usage: Event

Manual operation: See "Apply" on page 591

[SENSe<Ch>:]CORRection:COLLect:SCONnection<PhyPt> <TypeName>[, <Gender>]

For manual calibrations of channel <Ch>, this command defines or queries the VNA-side connector type and gender at (the calibration plane of) physical port <PhyPt>.

In contrast to `[SENSe<Ch>:]CORRection:COLLect:CONNection<PhyPt>`, it addresses the connector type by its name (ASCII string).

Suffix:

<Ch> Channel number of the calibrated channel

<PhyPt> Physical port number
If the analyzer is set to use the same connectors at all ports (`[SENSe<Ch>:]CORRection:COLLect:CONNection:PORTs ALL`), then a change of a connector type is valid for all ports. The gender of the connectors can still be different.

Parameters:

<TypeName> Connector type name (ASCII string)
Use `[SENSe:]CORRection:CONNection:CATalog?` to query for existing names.

*RST: The name of the current instrument's connector type.

<Gender> MALE | FEMale

Gender of the connectors. The gender designation is not necessary (and ignored) for sexless connector types.

Example: `*RST; :CORR:COLL:SCON1 'N 75 Ohm', MALE; SCON4?`
Change the connector type at port 1 from 'N 50 Ohm', FEM to 'N 75 Ohm', MALE. The connector type at the other ports is also changed to N 75 Ohm, however, the gender (female) is maintained. `CORR:COLL:SCON4?` returns `'N 75 Ohm', FEM`.

Manual operation: See "Connector / Port Gender" on page 586

[SENSe:]CORRection:COLLect:SWEep:CONTinuous <ContinuousSweep>

Disables/enables continuous calibration sweeps.

Parameters:

<ContinuousSweep> **ON (1)**
Continuous calibration sweeps enabled (system default)

OFF (0)
Continuous calibration sweeps disabled

Manual operation: See "Continuous Cal Sweeps" on page 814

7.3.14.7 [SENSe:]CORRection:METas

Using `[SENSe:]CORRection:METas` commands you can define additional properties of a METAS Calibration.

To perform a METAS calibration, use the standard commands for manual calibration (see Section 7.3.14.8, "[SENSe:]CORRection... (other)", on page 1375). For the calibration setup, use `[SENSe<Ch>:]CORRection:COLLect:METHod:DEFine` with METAS calibration types `MTOSm | MUOSm | MOSM | MTRL`.

- The related option R&S ZNB-K50 is only available for certain variants of the R&S ZNB. Contact your Rohde & Schwarz sales representative for more information.
- METAS calibration is not fully supported via remote control. The import of cal kits with uncertainties must be performed from the analyzer GUI, and must be done before the first METAS calibration.

[SENSe:]CORRection:METas:CABLe:CATalog? 1372
[SENSe<Ch>:]CORRection:METas:CABLe:MOVement 1372
[SENSe<Ch>:]CORRection:METas:CABLe:REConnection 1373
[SENSe<Ch>:]CORRection:METas:CABLe:TYPE 1373
[SENSe<Ch>:]CORRection:METas[:STATe] 1374
[SENSe<Ch>:]CORRection:METas:UNCertainty 1374

[SENSe:]CORRection:METas:CABLe:CATalog?

Lists the available METAS cable models, identified by their file names (without `*.cable` extension) in the local METAS VNA Tools II database.

Usage: Query only

Options: R&S ZNB-K50

Manual operation: See "Cable" on page 664

[SENSe<Ch>:]CORRection:METas:CABLe:MOVement [<Ports>[, <Ports>]...]

While a METAS calibration is active for the related channel (see `[SENSe<Ch>:]CORRection:METas[:STATe]` on page 1374), this command notifies the METAS VNA Tools II about additional cable movements at the specified ports.

Suffix:
<Ch> Channel number

Setting parameters:
<Ports> **<integer>**
First port number, where cables were moved

ALL
Cable movements at all calibrated ports

NONE
Clears the cable movement history in the METAS project

<Ports> Additional port numbers, where cables were moved

Usage: Setting only

Options: R&S ZNB-K50

Manual operation: See "Add New Connection/Add Cable Movement" on page 670

[SENSe<Ch>:]CORRection:METas:CABLe:REConnection [<Ports>[, <Ports>]...]

While a METAS calibration is active for the related channel (see `[SENSe<Ch>:]CORRection:METas[:STATe]` on page 1374), this command notifies the METAS VNA Tools II about additional connector reconnections at the specified ports.

Suffix:
<Ch> Channel number

Setting parameters:
<Ports> **<integer>**
First port number, where connectors were reconnected

ALL
All ports were reconnected

NONE
Clears the connector reconnection history in the METAS project

<Ports> Additional port numbers, where connectors were reconnected

Usage: Setting only

Options: R&S ZNB-K50

Manual operation: See "Add New Connection/Add Cable Movement" on page 670

[SENSe<Ch>:]CORRection:METas:CABLe:TYPE <CableNameFile>[, <Ports>[, <Ports>]...]

Selects the METAS cable type that is used at the respective ports.

Use `[SENSe:]CORRection:METas:CABLe:CATalog?` to list the available cable types.

Suffix:
<Ch> Channel number
The related channel must be prepared for a METAS calibration using `[SENSe<Ch>:]CORRection:COLLect:METHod:DEFine<CalName>, MTOSm | MUOSm | MOSM | MTRL, ....`

Setting parameters:

<CableNameFile> Name of a METAS cable model file (without `*.cable` extension) in the database of the local METAS VNA Tools II installation.

<Ports> First port number or `ALL`, if the same cable type is used for all ports.
The latter is equivalent to omitting all <Ports> parameters.

<Ports> Additional port numbers

Usage: Setting only

Options: R&S ZNB-K50

Manual operation: See "Cable" on page 664

[SENSe<Ch>:]CORRection:METas[:STATe] <Boolean>

After a successful METAS calibration, this command activates or deactivates the uncertainty calculation for the related channel.

Suffix:

<Ch> Channel number

Parameters:

<Boolean> **ON (1)**
METAS calibration active
OFF (0)
METAS calibration inactive

Options: R&S ZNB-K50

Manual operation: See "METAS Cal Active" on page 661

[SENSe<Ch>:]CORRection:METas:UNCertainty <Level>

Defines the width of the uncertainty band, expressed as multiples of the standard deviation.

Default is 2 (95% confidence interval).

Suffix:

<Ch> Channel number

Parameters:

<Level> Scalar value between 1 and 10
Increment: 0.1
*RST: 2

Options: R&S ZNB-K50

Manual operation: See "Significance" on page 663

7.3.14.8 [SENSe:]CORRection... (other)

The remaining `[SENSe:]CORRection...` commands.

[SENSe<Ch>:]CORRection:CDATa....1376
[SENSe<Ch>:]CORRection:CDATa:PORT<PhyPt>....1376
[SENSe:]CORRection:CONNection....1378
[SENSe:]CORRection:CONNection:CATalog?....1380
[SENSe<Ch>:]CORRection:CONNection:DELete....1380
[SENSe<Ch>:]CORRection:DATA:PARameter<Sfk>?....1380
[SENSe<Ch>:]CORRection:DATA:PARameter<Sfk>:PORT<PhyPt>?....1380
[SENSe<Ch>:]CORRection:DATA:PARameter:COUNt?....1383
[SENSe<Ch>:]CORRection:DATE?....1383
[SENSe<Ch>:]CORRection:DELete....1384
[SENSe<Ch>:]CORRection:EDELay:AUTO....1384
[SENSe<Ch>:]CORRection:EDELay<PhyPt>:DIELectric....1384
[SENSe<Ch>:]CORRection:EDELay<PhyPt>:DISTance....1384
[SENSe<Ch>:]CORRection:EDELay<PhyPt>:ELENgth....1385
[SENSe<Ch>:]CORRection:EDELay<PhyPt>[:TIME]....1385
[SENSe<Ch>:]CORRection:EDELay<PhyPt>:VELocity....1386
[SENSe:]CORRection:EDELay:VNETwork....1386
[SENSe<Ch>:]CORRection:LOSS<PhyPt>....1386
[SENSe<Ch>:]CORRection:LOSS<PhyPt>:SECond....1386
[SENSe<Ch>:]CORRection:LOSS:AUTO....1387
[SENSe<Ch>:]CORRection:LOSS<PhyPt>:FREQuency....1387
[SENSe<Ch>:]CORRection:LOSS<PhyPt>:SECond:FREQuency....1387
[SENSe<Ch>:]CORRection:LOSS<PhyPt>:OFFSet....1388
[SENSe<Ch>:]CORRection:LOSS<PhyPt>:SECond:STATe....1388
[SENSe<Chn>:]CORRection:NSTate?....1388
[SENSe<Ch>:]CORRection:OFFSet<PhyPt>:COMPensation[:STATe]....1389
[SENSe<Ch>:]CORRection:OFFSet<PhyPt>:DFComp[:STATe]?....1389
[SENSe<Ch>:]CORRection:OFFSet<PhyPt>[:STATe]....1389
[SENSe<Ch>:]CORRection:PCAL....1390
[SENSe<Ch>:]CORRection:POWer<PhyPt>:ACQuire....1391
[SENSe<Ch>:]CORRection:POWer<PhyPt>:AWAVe[:STATe]....1392
[SENSe<Ch>:]CORRection:POWer:DATA....1392
[SENSe<Ch>:]CORRection:POWer:DATA:PORT<PhyPt>....1392
[SENSe<Ch>:]CORRection:POWer<PhyPt>:IMODulation:ACQuire....1393
[SENSe<Ch>:]CORRection:POWer<PhyPt>[:STATe]....1394
[SENSe<Chn>:]CORRection:PSTate?....1395
[SENSe<Ch>:]CORRection:SMATrix:CDATa....1395
[SENSe<Ch>:]CORRection:SMATrix:CDATa:PORT<PhyPt>....1395
[SENSe<Ch>:]CORRection[:STATe]....1396
[SENSe<Ch>:]CORRection:STIMulus?....1396
[SENSe<Ch>:]CORRection:STIMulus:PORT<PhyPt>?....1396
[SENSe<Chn>:]CORRection:SSTate?....1397

[SENSe<Ch>:]CORRection:CDATa <ErrorTerm>, <SourcePort>, <LoadPort>, <CorrectionData>...
[SENSe<Ch>:]CORRection:CDATa:PORT<PhyPt> <ErrorTerm>, <SourcePort>, <LoadPort>, <CorrectionData>...

Writes or reads system error correction data for a specific channel `<Ch>`, calibration method (`[SENSe<Ch>:]CORRection:COLLect:METHod:DEFine`), and port combination `<SourcePort>`, `<LoadPort>`. The setting command can be used to transfer user-defined correction data `<CorrectionData>` to the analyzer; the query returns the current correction data set. ASCII or block data can be transferred, depending on the selected data transfer format (`FORMat[:DATA]`).

The sweep must be stopped to transfer calibration data; see program example for `[SENSe<Ch>:]CORRection:COLLect:SAVE:SELected:DEFault`.

Note: This command affects the active calibration of channel no. `<Ch>` or the factory calibration (if no channel calibration is active). For the factory calibration, the query form is allowed only (no change of factory calibration data).

Tip: The analyzer provides a default calibration corresponding to a test setup which does not introduce any systematic errors; see `[SENSe<Ch>:]CORRection:COLLect:SAVE:SELected:DEFault`.

For an overview of calibration methods and error terms, refer to Section 4.5.1, "Calibration types", on page 155.

G and H matrices

The 7-term calibration types named Txx (e.g. TOM, TRM, TRL, TNA) are based on a network analyzer with two ports i and j, each equipped with a test receiver and a reference receiver. The system errors are described in terms of two "error two-ports" P_G and P_H:

- The error two-port P_G is assigned to port i of the analyzer. Its transmission matrix G describes how the system errors modify the outgoing and incident waves at port i:
$$\begin{bmatrix} b_i \\ a_i \end{bmatrix} = \begin{bmatrix} G_{11} & G_{12} \\ G_{21} & G_{22} \end{bmatrix} * \begin{bmatrix} m_{i\,ref} \\ m_{i\,test} \end{bmatrix}$$
- The error two-port P_H is assigned to port j of the analyzer. Its transmission matrix H describes how the system errors modify the measured incident and outgoing waves at port j:
$$\begin{bmatrix} a_j \\ b_j \end{bmatrix} = \begin{bmatrix} H_{11} & H_{12} \\ H_{21} & H_{22} \end{bmatrix} * \begin{bmatrix} m_{j\,test} \\ m_{j\,ref} \end{bmatrix}$$

In the two equations above, a and b denote the waves at the calibrated reference plane i and j (e.g. the input and output of the 2-port DUT). The m waves are the raw measured waves of test port i and j. The subscripts "ref" and "test" refer to the reference and test receivers, respectively. During the calibration, the network analyzer acquires ratios of wave quantities, which leaves one of non-diagonal matrix elements of G or H as a free normalization factor. The network analyzer uses the normalization $H_{21} = 1$.

Suffix:

<Ch> Channel number of the calibrated channel.

<PhyPt> Physical port number.
Used to select a specific frequency axis in arbitrary mode (with option R&S ZNB-K4).

Parameters:

<ErrorTerm> String parameters describing the different error terms, depending on the current calibration method; see table below. Each term contains one complex value (real and imaginary part) for each sweep point. The parameters must be transferred in full length. The following strings are allowed:

'DIRECTIVITY'
Directivity at source port <SourcePort>

'SRCMATCH'
Source match at source port <SourcePort>

'REFLTRACK'
Reflection tracking at source port <SourcePort>

'LOADMATCH'
Load match at load port <LoadPort>

'TRANSTRACK'
Transmission tracking between source port <SourcePort> and load port <LoadPort>.

'G11' ... 'G22'
G matrix elements at source port <SourcePort>; see explanation above.

'H11' ... 'H22'
H matrix elements at load port <LoadPort>; see explanation above.

'Q11' ... 'Q22 ', 'PREL'
9-term correction factors, defined at source port <SourcePort>; <LoadPort> is ignored.

‘PABS’
9-term correction factors, defined for the lowest source port <SourcePort> for each SMARTerCal; <LoadPort> is ignored

'L1', 'L2'
9-term correction factors, defined between the source port <SourcePort> and the load port <LoadPort>

Range: The error terms are dimensionless complex numbers.
*RST: n/a

<SourcePort> Source port number

<LoadPort> Load port number. If the error term is not related to the load port, a dummy number can be used; e.g. `CORR:CDAT 'REFLTRACK', 1, 0`

<CorrectionData> Correction data set (one complex number per sweep point) to be transferred to the analyzer either in ASCII or block data format, depending on the current `FORMat[:DATA]` setting. The correction data set is assigned to the specified channel, error term, source and load port.
Do not use this parameter for queries.

Example: See `[SENSe<Ch>:]CORRection:COLLect:SAVE:SELected:DEFault`.

The different calibration types of the analyzer provide the following error terms:

Calibration type	Parameters in `[SENSe<Ch>:]CORRection:COLLect:METHod:DEFine`	Available error terms (depending on port numbers)
One-port normalization (reflection) using an open or a short standard	REFL, RSHort	'REFLTRACK'
Full one port ("Refl OSM")	FOPort	'DIRECTIVITY', 'SRCMATCH', 'REFLTRACK'
Two-port normalization	FRTRans	'TRANSTRACK'
One path two port	OPTPort	'DIRECTIVITY', 'SRCMATCH', 'REFLTRACK', 'TRANSTRACK'
TOSM	TOSM	'DIRECTIVITY', 'SRCMATCH', 'REFLTRACK', 'LOADMATCH', 'TRANSTRACK'
TOM, TSM, TRM, TRL, TNA	TOM | TSM | TRM | TRL | TNA	'DIRECTIVITY', 'SRCMATCH', 'REFLTRACK', 'LOADMATCH', 'TRANSTRACK' (for reading and writing) 'G11' ... 'G22' and 'H11', 'H12', 'H22' (for reading only; the 'H21' matrix elements are normalized to 1)

[SENSe:]CORRection:CONNection <ConnectorName>[, <Mode>[, <Gender>[, <RelPermittivity>[, <Impedance or Cutoff>]]]]

The setting defines a custom connector type. All parameters are required.

The query `[SENSe:]CORRection:CONNection? <ConnectorName>` returns the properties of connector type `<ConnectorName>`.

Parameters:

<ConnectorName> Non-empty ASCII string
Use `[SENSe:]CORRection:CONNection:CATalog?` to get a list of the existing connector names.

Reserved names: If you define connectors named "UserConn1" and "UserConn2", you can use the predefined constants `USER1` and `USER2` to refer to these connector types in the ZVR-style command `[SENSe:]CORRection:CKIT:<ConnType>:SELect`. Similarly, you can use the constants `UFEMale1 | UMALe1 | UFEMale2 | UMALe2` to refer to the male/female variants of these connector types in the ZVR-style command `[SENSe<Ch>:]CORRection:COLLect:CONNection<PhyPt>`.
Using "Ohm" in connector names: At the VNA **GUI**, the string "Ohm" (case-insensitive) is represented by the symbol "Ω". Conversely, if you define a cal connector type at the GUI with an Ω symbol in its name, the "Ω" is converted to the string "Ohm" at the remote interface.

<Mode>
TEM | WGUide
Propagation mode
TEM
Transverse electromagnetic (TEM) modes
WGUide
Waveguide modes

<Gender>
GENDer | NGENder
GENDer
Polar connector type (m/f)
NGENder
Sexless connector type

<RelPermittivity>
Relative permittivity
Range: 0.0000000001 to 1000.

<Impedance or Cutoff>
For **TEM** propagation <Mode>, the characteristic impedance in Ω (without unit).
For **WGUide** propagation <Mode>, the cutoff frequency in Hz (without unit).
Range: Ref. impedance: 1µΩ to 1000 MΩ. Cutoff frequency: 0 Hz to 1000 GHz.
*RST: -

Example:
```
CORR:CONN 'USERCON',TEM,GEND,1.00000,50
```
Define a TEM type connector type named `USERCON`.
```
CORR:CONN? 'USERCON'
```
Query the properties of the configured connector type.
```
CORR:CONN:DEL 'USERCON'
```
Delete the configured connector type.

Manual operation: See "Connector / Port Gender" on page 586

[SENSe:]CORRection:CONNection:CATalog?

Returns a comma-separated list of all cal connector type names.

Use `[SENSe:]CORRection:CONNection` to create custom cal connector types.

Example: See `[SENSe:]CORRection:CKIT:CATalog?`.

Usage: Query only

Manual operation: See "Connector / Port Gender" on page 586

[SENSe<Ch>:]CORRection:CONNection:DELete <ConnectorName>

Deletes a user-defined connector type named `<ConnectorName>`.

Suffix:
<Ch> Channel number

Setting parameters:
<ConnectorName> Name of the user-defined connectors, string parameter.

Example: See `[SENSe:]CORRection:CONNection`

Usage: Setting only

Manual operation: See "Connector / Port Gender" on page 586

[SENSe<Ch>:]CORRection:DATA:PARameter<Sfk>? [<SfkSettingType>[, <Index>]]
[SENSe<Ch>:]CORRection:DATA:PARameter<Sfk>:PORT<PhyPt>?
[<SfkSettingType>[, <Index>]]

Gets the settings of active system error correction no. <Sfk> (or of the factory calibration, if no channel calibration is active).

Suffix:
<Ch> Number of the calibrated channel.

<Sfk> Number of the system error correction.
Less or equal than the total number of active system error corrections for the related channel (see `[SENSe<Ch>:]CORRection:DATA:PARameter:COUNt?` on page 1383).

<PhyPt> Physical port number, used to select a specific frequency axis in arbitrary mode (with option R&S ZNB-K4).

Query parameters:
<SfkSettingType> ACAL | STARt | STOP | POINts | SPOWer | STYPe | BANDwidth | PDLY | RATTenuation | TYPE | PORTs | SPORt | THRoughs | TSTamp | LTSTamp | TVNA | MVNA | MTESt | CKIT | FSMode

The requested setting.
If no <SfkSettingType> is specified, the values for STARt, STOP, POINts, SPOWer and STYPe are returned

ACAL
1 for automatic calibrations, 0 for manual calibrations

STARt
Start frequency (or CW frequency, if no frequency sweep is active)

STOP
Stop frequency (or CW frequency, if no frequency sweep is active)

POINts
Number of points

SPOWer
Source power (or stop power for power sweeps)

STYPe
Sweep type or grid (LIN, LOG, SEGM)

BANDwidth
Measurement bandwidth

PDLY
Point delay (Meas Delay) for OSM

RATTenuation
Receiver attenuations: comma-separated list containing a value pair <port no.>,<rec. att.> (float,integer) for each of the involved **PORTs**

TYPE
Calibration type (see <Type> parameter in `[SENSe<Ch>:]CORRection:COLLect[:ACQuire]:SELected` on page 1331)

PORTs
The related test port numbers (comma-separated list of integers)

SPORt
For SMARTer calibrations this indicates the port to which a powermeter was connected

THRoughs
Measued Throughs: comma-separated list of directed port number pairs <pno1-pno2>

TSTamp
Timestamp in UTC

LTSTamp
Timestamp in local time

TVNA
For calibrations involving switch matrices, this indicates the test ports on the VNA itself. Returns a comma-separated list of port pairs `TestPort,VnaPort`

MVNA
For calibrations involving switch matrices, this indicates the mapping between matrix ports and VNA ports for matrix <Index>. Returns a comma-separated list of port pairs
`MatrixVnaPort,VnaPort`

MTESt
For calibrations involving switch matrices, this indicates the mapping between matrix (physical) test ports and test ports for matrix <Index>. Returns a comma-separated list of port pairs
`MatrixTestPort,TestPort`

CKIT
If created with FW version 2.30 or higher, for each port the name of the used calibration kit is stored with a calibration.
The query can return one of the following:
the cal kit name (string)
if available and unique
'Multiple'
if cal kit names are available but not unique, i.e. if <PhyPt> was not specified and multiple cal kits were used (multi-port cals with different connector types, merged cals)
'Unknown'
if cal kit names are not available (cals created with a FW version < 2.30)

FSMode
Returns the frequency sweep mode that was used during calibration.
STEP: Stepped mode (for all segments)
ANAL: Swept mode (for at least one segment)
See Section 4.1.4.5, "Stepped vs. swept mode", on page 88.

<Index> If one or more external switch matrices were used during calibration, this refers to the index of a particular switch matrix (see `SYSTem:COMMunicate:RDEVice:SMATrix<Matr>:DEFine` on page 1559).

Example:
```
SENSe:CORRection:DATA:PARameter?
```
Unrestricted query. Result looks like this:
100000,8500000000,201,-10,LIN
```
SENSe:CORRection:DATA:PARameter? STARt
...
SENSe:CORRection:DATA:PARameter? STYPe
```
Query settings one by one.
```
SENSe:CORRection:DATA:PARameter? RATTenuation
```
Query receiver attenuations. Result looks like this:
1,0.000000,2,0.000000

Usage: Query only

Manual operation: See "Apply" on page 591

[SENSe<Ch>:]CORRection:DATA:PARameter:COUNt?

Gets the number of active system error corrections for channel <Ch>.

This number can be > 1 in case the enhanced multi-calibration wizard was used (see `SYSTem:DISPlay:DIALogs:SETup:MCAL[:STATe]` on page 1568) to collect the calibration data. The properties of the active system error corrections can be queried using `[SENSe<Ch>:]CORRection:DATA:PARameter<Sfk>?`.

Suffix:
<Ch> Channel number

Usage: Query only

Manual operation: See "Ch<n> Calibration Properties/Cal Group <n> Properties" on page 659

[SENSe<Ch>:]CORRection:DATE?

Returns the date and time when the active system error correction data for channel `<Ch>` was acquired (see example).

Suffix:
<Ch> Channel number of the calibrated channel

Example:
```
CORR:COLL:METH REFL1
```
Select a one-port normalization at port 1 as calibration type.
```
CORR:COLL OPEN1
```
Measure an open standard connected to port 1 and store the measurement results of this standard.
```
CORR:COLL:SAVE
```
Calculate the system error correction data and apply it to the active channel.
```
CORR:DATE?
```
Query the time when the system error correction became active. The analyzer returns the date and time, e.g.
`'03/20/11,18:30:39'`.
```
CORR:DATA:PAR?
```
Query the sweep settings for the calibration sweep. The analyzer returns the start and stop frequency, the number of points, source power, and the sweep type, e.g.
`300000,8000000000,201,0,LIN`.
```
CORR:SST?
```
Query the calibration status. The analyzer returns `'CAL OFF'` (because the performed one-port calibration is not sufficient for the measured transmission S-parameter S_{21}).

Usage: Query only

Manual operation: See "Apply" on page 591

[SENSe<Ch>:]CORRection:DELete

Cleans the channel's existing calibration data and also removes links to the calibration pool.

Suffix:
<Ch> Channel number

Usage: Event

Manual operation: See "Manage Cals – Remove all Cal Data" on page 657

[SENSe<Ch>:]CORRection:EDELay:AUTO <Activate>

Applies the Auto Length) function to the active trace of channel <Ch>.

Use `CALCulate<Chn>:STATistics:DOMain:USER` commands to limit the frequency range for the calculation.

Suffix:
<Ch> Channel number

Setting parameters:
<Activate> ONCE
Applies the auto length function.

Example:
```
*RST; :CORR:EDEL:AUTO ONCE
```
Reset the instrument and apply the auto length function to the default trace (Trc1 in channel 1).

Usage: Setting only

Manual operation: See "Auto Length" on page 740

[SENSe<Ch>:]CORRection:EDELay<PhyPt>:DIELectric <Permittivity>

Defines the permittivity for the offset correction at test port `<PhyPt>`.

Suffix:
<Ch> Channel number of the offset-corrected channel

<PhyPt> Port number of the analyzer

Parameters:
<Permittivity> Permittivity
Range: 1 to +1E+6
*RST: 1.00062

Example: See `[SENSe<Ch>:]CORRection:EDELay<PhyPt>:ELENgth`

Manual operation: See "Permittivity / Velocity Factor" on page 739

[SENSe<Ch>:]CORRection:EDELay<PhyPt>:DISTance <MechLength>

Defines the offset parameter for test port `<PhyPt>` as a mechanical length.

Suffix:
<Ch> Channel number of the offset-corrected channel

<PhyPt> Port number of the analyzer

Parameters:
<MechLength> Mechanical length

Range: -3.402823466E+038 m to +3.4028234664E+038 m.
*RST: 0 m
Default unit: m

Example: See `[SENSe<Ch>:]CORRection:EDELay<PhyPt>:ELENgth`

Manual operation: See "Delay / Electrical Length / Mech. Length" on page 738

[SENSe<Ch>:]CORRection:EDELay<PhyPt>:ELENgth <ElecLength>

Defines the offset parameter for test port `<PhyPt>` as an electrical length.

Suffix:
<Ch> Channel number of the offset-corrected channel

<PhyPt> Port number of the analyzer

Parameters:
<ElecLength> Electrical length

Range: -1E+9 m to +1E+9 m.
Increment: 1 mm
*RST: 0 m
Default unit: m

Example:
```
CORR:EDEL2:ELEN 0.3
```
Define an electrical length of 30 cm for channel 1 and port no. 2.
```
CORR:EDEL2:DIST?; DIEL?
```
Query the values of the mechanical length and the permittivity at port 2. The mechanical length is equal to the electrical length divided by the square root of the permittivity; the latter is set to its default value. The response is `0.29990704322;1.00062`.
```
CORR:EDEL2?
```
Query the value of the delay at port 2. The delay is equal to the electrical length divided by the speed of light in the vacuum, so the response is `1.0006922856E-009`.
```
CORR:LOSS2 2; LOSS2:FREQ 1.5 GHz; OFFS 3 dB
```
Define the offset loss parameters at port 2.

Manual operation: See "Delay / Electrical Length / Mech. Length" on page 738

[SENSe<Ch>:]CORRection:EDELay<PhyPt>[:TIME] <Delay>

Defines the offset parameter for test port `<PhyPt>` as a delay time.

Suffix:
<Ch> Channel number of the offset-corrected channel

<PhyPt> Port number of the analyzer

Parameters:

<Delay> Delay

Range: -3.40282346638529E+038 s to +3.40282346638529E+038 s.

*RST: 0 s

Default unit: s

Example: See `[SENSe<Ch>:]CORRection:EDELay<PhyPt>:ELENgth`

Manual operation: See "Delay / Electrical Length / Mech. Length" on page 738

[SENSe<Ch>:]CORRection:EDELay<PhyPt>:VELocity <VelocityFactor>

Defines the velocity factor for the offset correction at test port `<PhyPt>`.

Suffix:

<Ch> Channel number of the offset-corrected channel

<PhyPt> Port number of the analyzer

Parameters:

<VelocityFactor> Velocity factor

Range: 0.01 to 1

Manual operation: See "Permittivity / Velocity Factor" on page 739

[SENSe:]CORRection:EDELay:VNETwork <Boolean>

Changes the position of the offset calculation in the "Offset Embed" calculation chain.

Parameters:

<Boolean> **OFF (0)**
The offset is calculated **before** de-/embedding (default).

ON (1)
The offset is calculated **after** de-/embedding.

*RST: 0

Manual operation: See "Offset > Calculate after De-/Embed." on page 760

[SENSe<Ch>:]CORRection:LOSS<PhyPt> <LossAtFrequency>
[SENSe<Ch>:]CORRection:LOSS<PhyPt>:SECond <LossAtFrequency>

These commands define the offset loss at the reference frequencies f_1 and f_2.

The reference frequencies can be set using `[SENSe<Ch>:]CORRection:LOSS<PhyPt>:FREQuency` and `[SENSe<Ch>:]CORRection:LOSS<PhyPt>:SECond:FREQuency`.

Suffix:

<Ch> Channel number

<PhyPt> Physical port number

Parameters:

<LossAtFrequency> Loss at reference frequency

Range: -200 dB to +200 dB
Increment: 0.001 dB
*RST: 0 dB
Default unit: dB

Manual operation: See "Loss at DC / Loss at 1st Freq / 1st Freq for Loss / Use 2nd Freq / Loss at 2nd Freq / 2nd Freq for Loss" on page 706

[SENSe<Ch>:]CORRection:LOSS:AUTO <Activate>

Applies the Auto Length and Loss) function to the active trace of channel <Ch>.

Use `CALCulate<Chn>:STATistics:DOMain:USER` commands to limit the frequency range for the calculation.

Suffix:

<Ch> Channel number

Setting parameters:

<Activate> ONCE

Required constant

Example:

```
*RST; :CORR:LOSS:AUTO ONCE
```

Reset the instrument and apply the "Auto Length and Loss" function to the default trace (Trc1 in channel 1).

Usage: Setting only

Manual operation: See "Auto Length and Loss" on page 744

[SENSe<Ch>:]CORRection:LOSS<PhyPt>:FREQuency <RefFreq>
[SENSe<Ch>:]CORRection:LOSS<PhyPt>:SECond:FREQuency <RefFreq>

These commands define the reference frequencies f_1 and f_2 for the offset loss approximation.

The corresponding losses can be defined using `[SENSe<Ch>:]CORRection:LOSS<PhyPt>:OFFSet`.

Suffix:

<Ch> Channel number

<PhyPt> Physical port number

Parameters:

<RefFreq> Reference frequency

Range: Frequency range of the analyzer model.
Increment: 1 MHz
*RST: 1000000000 Hz (= 1 GHz)
Default unit: Hz

Manual operation: See "Loss at DC / Loss at 1st Freq / 1st Freq for Loss / Use 2nd Freq / Loss at 2nd Freq / 2nd Freq for Loss" on page 706

[SENSe<Ch>:]CORRection:LOSS<PhyPt>:OFFSet <OffsetLoss>

Defines the frequency-independent part (DC value) of the offset loss.

Suffix:
<Ch> Channel number of the offset-corrected channel

<PhyPt> Port number of the analyzer

Parameters:
<OffsetLoss> Frequency-independent part of the offset loss

Range: -200 dB to +200 dB
Increment: 0.001 dB
*RST: 0 dB
Default unit: dB

Example: See `[SENSe<Ch>:]CORRection:EDELay<PhyPt>:ELENgth`

Manual operation: See "Loss at DC / Loss at 1st Freq / 1st Freq for Loss / Use 2nd Freq / Loss at 2nd Freq / 2nd Freq for Loss" on page 706

[SENSe<Ch>:]CORRection:LOSS<PhyPt>:SECond:STATe <Boolean>

Defines whether the second reference frequency f_2 is used for the offset loss approximation.

The second reference frequency and its associated loss value can be defined using `[SENSe<Ch>:]CORRection:LOSS<PhyPt>:SECond:FREQuency` and `[SENSe<Ch>:]CORRection:LOSS<PhyPt>:SECond`.

Suffix:
<Ch> Channel number

<PhyPt> Physical port number

Parameters:
<Boolean>

OFF (0)
f_2 not used (default)

ON (1)
f_2 used

Manual operation: See "Loss at DC / Loss at 1st Freq / 1st Freq for Loss / Use 2nd Freq / Loss at 2nd Freq / 2nd Freq for Loss" on page 706

[SENSe<Chn>:]CORRection:NSTate?

Queries the noise figure calibration state label of the active trace in channel <Chn>. The active trace must be a noise figure trace, otherwise an empty string is returned.

Suffix:
<Chn> Channel number used to identify the active trace

Usage: Query only

Options: R&S ZNB-K30

[SENSe<Ch>:]CORRection:OFFSet<PhyPt>:COMPensation[:STATe] <Boolean>

Toggles length/loss/fixture compensation for physical port <PhyPt> ON/OFF.

Suffix:
<Ch> Channel number

<PhyPt> Physical port number

Parameters:
<Boolean> ON (1): compensation active
OFF (0): compensation inactive

Manual operation: See "Active" on page 706

[SENSe<Ch>:]CORRection:OFFSet<PhyPt>:DFComp[:STATe]?

Returns whether a direct fixture compensation has been carried out at port no. `<PhyPt>`. A direct fixture compensation resets the offset parameters to zero, the analyzer uses calculated transmission factors instead.

Suffix:
<Ch> Channel number of the offset-corrected channel

<PhyPt> Port number of the analyzer.

Return values:
<Boolean> 1 - direct fixture compensation data used
0 - no direct fixture compensation data used

Example: `*RST; CORR:OFFS:DFC?`
Reset the instrument and query whether the analyzer uses direct fixture compensation data at port 1. The response is 0.

Usage: Query only

Manual operation: See "Direct Compensation" on page 742

[SENSe<Ch>:]CORRection:OFFSet<PhyPt>[:STATe] <Boolean>

Resets the offset parameters for all test ports to zero and the reference frequency to 1 GHz or queries whether any of the offset parameters are different from zero.

Suffix:
<Ch> Channel number of the offset-corrected channel

<PhyPt> Port number of the analyzer. This numeric suffix is ignored; the command affects the parameters of all ports.

Parameters:

<Boolean> The parameter function depends on whether the command is used as a setting command or as a query: For setting command:
ON - no effect
OFF - resets all length offsets to zero and the reference frequency to 1 GHz
For query:
1 - at least one length offset parameter is different from its default value
0 - all length offsets are zero / set to default

*RST: OFF

Example: `*RST; :CORR:OFFS?`
Reset the instrument and query whether the length offset parameters have been reset as well. The response is 0.

Manual operation: See "Reset Offsets" on page 707

[SENSe<Ch>:]CORRection:PCAL <ActivatePowerCals>

Activates or deactivates all power calibrations.

Suffix:

<Ch> Calibrated channel number

Setting parameters:

<ActivatePowerCals> NONE | ALL

NONE – deactivate all power calibrations.
ALL – activate all power calibrations.

*RST: n/a

Example: `*RST; CORR:POW:ACQ AWAVe,1,PORT,1`
Perform a receiver power calibration of the wave a_1 using port 1 as a source port, assuming that the transmitted source power a_1 is correct. No external cabling is needed.
`CORR:POW:AWAV?`
Check whether the calibration is applied (the response is `1`).
`CORR:PCAL NONE; POW:AWAV?`
Disable all power calibrations and check again whether the calibration is applied (the response is `0`).

Usage: Setting only

Manual operation: See "Scalar Power Cal – All Power Cals On / All Power Cals Off" on page 656

[SENSe<Ch>:]CORRection:POWer<PhyPt>:ACQuire <Wave>[, <CalPort>[, <SourceType>[, <SourcePort>[, <CalOnlyPortFreq>]]]]

Selects the wave quantity and the source for the receiver power calibration, starts the calibration sweep, and applies the receiver power correction.

Suffix:

<Ch> Calibrated channel number

<PhyPt> This suffix is ignored because the port number is specified in the parameter list.

Setting parameters:

<Wave> AWAVe | BWAVe | B1 | B2 | B3 | B4

AWAVe – calibration of reference waves a_1, a_2, ... In manual control, the reference receiver calibration is included in the source power calibration of each port.
BWAVe – calibration of received wave b_1, b_2, ... The port number `<CalPort>`, the used `<SourceType>`, and the `<SourcePort>` number must be specified in addition.
Alternative: Parameters B1 and B2.
B1 | B2 ... – direct wave and cal port setting for received waves b_1, b_2 ... The parameters B1 and B2 are ZVR-compatible. No additional parameters must be specified. The source for B1 is port 2 and vice versa.

<CalPort> Calibrated port number

Range: 1 to port number of the analyzer

<SourceType> PORT | GENerator

PORT - internal source at port `<CalPort>`, to be fed to port `<SourcePort>` using an external through connection. If `<CalPort> = <SourcePort>`, an Open or Short standard is required.
GENerator - external generator no. <source_no>.

<SourcePort> Number of the port for the internal source or generator.

<CalOnlyPortFreq> Relevant for frequency conversion measurements only.

OFF | 0
The reference receiver at port <CalPort> is calibrated for all frequencies that are relevant for any of the ports.

ON | 1
Only those frequencies are calibrated that are relevant for this port. This results in shorter calibration times.

*RST: 0

Example: `*RST; CORR:POW:ACQ AWAVe,1,PORT,1`
Perform a receiver power calibration of the wave a_1 using port 1 as a source port, assuming that the transmitted source power a_1 is correct. No external cabling is needed.
`CORR:POW:AWAV?`
Check whether the calibration is applied (the response is `1`).
`CALC:PAR:MEAS 'TRC1', 'A1'`
Select a_1 as a measured quantity for the default trace.
`CALC:MARK ON; MARK:Y?; :SOUR:POW?`
Create marker no. `<Mk>` in the center of the sweep range and query the measurement value. The calibrated power of the reference wave a_1 is approx. equal to the default source power value.
See also example for `[SENSe<Ch>:]CORRection:OFFSet<PhyPt>[:STATe]`.

Usage: Setting only

Manual operation: See "Port Overview" on page 597

[SENSe<Ch>:]CORRection:POWer<PhyPt>:AWAVe[:STATe] <Boolean>

Enables or disables the receiver power calibration for channel `<Ch>` and for the reference waves a_n. The setting command is disabled unless the reference waves have been power calibrated (`[SENSe<Ch>:]CORRection:POWer<PhyPt>:ACQuire` on page 1391 `AWAVe,...`). The query always returns a result.

This command extends the functionality of manual control, where the reference receiver calibration is included in the source power calibration of each port (and always ON).

Suffix:
<Ch> Calibrated channel number

<PhyPt> Calibrated port number

Parameters:
<Boolean> Enables (ON) or disables (OFF) the receiver power calibration for the reference waves a_n.

*RST: OFF

Example: See `[SENSe<Ch>:]CORRection:POWer<PhyPt>:ACQuire`

[SENSe<Ch>:]CORRection:POWer:DATA <Wave>, <CorrData>...
[SENSe<Ch>:]CORRection:POWer:DATA:PORT<PhyPt> <Wave>, <CorrData>...

Reads or writes receiver power correction data sets. A power correction data set contains n real values where:

- Each value corresponds to the ratio of the actual power at the receiver input (value provided by the used source) to the uncalibrated power in dB.
- The number n is equal to the number of sweep points.

Increasing (decreasing) the values in the correction data sets increases (decreases) the input power reading. Writing correction data (the setting command) fails if the number of transferred values is not equal to the number of sweep points.

Suffix:

<Ch> Calibrated channel number

<PhyPt> Physical port number, used to select a specific frequency axis in arbitrary mode (with option R&S ZNB-K4).

Parameters:

<Wave> Identifier for the calibrated wave:
'A<n>' or 'AWAV<n>' denote correction data for the reference wave a<n>, where <n> corresponds to the port number.
'B<n>' or 'BWAV<n>' denote correction data for the reference wave b<n>.

<CorrData> Power correction values either in ASCII or block data format, depending on the current `FORMat[:DATA]` setting.

Example:

```
*RST; :SWE:POIN 10
```
Reset the instrument and reduce the number of sweep points to 10.
```
CORR:POW:ACQ AWAVe,1,PORT,1
```
Perform a receiver power calibration of the wave a_1 using port 1 as a source port, assuming that the transmitted source power a_1 is correct. No external cabling is needed.
```
CORR:POW:DATA? 'AWAV1'
```
Query the correction values. The analyzer returns 10 comma-separated real numbers.
```
CORR:POW:DATA 'AWAV1', 1, 2, 3, 4, 5, -6, -7, -8, -9, -0
```
Replace the correction values by 10 (new) numbers.

Manual operation: See "Start Cal Sweep" on page 600

[SENSe<Ch>:]CORRection:POWer<PhyPt>:IMODulation:ACQuire
[<CalOnlyPortFreq>]

Starts the receiver calibration (3rd power calibration step for intermodulation measurements), stores and applies the calibration data.

Note: The receiver calibration relies on the source power calibration acquired in step nos. 1 and 2.
`[SENSe<Ch>:]CORRection:POWer<PhyPt>:IMODulation:ACQuire` is enabled only after steps 1 and 2 have been carried out (`SOURce<Ch>:POWer<PhyPt>:CORRection:IMODulation:LTONe[:ACQuire]`, `SOURce<Ch>:POWer<PhyPt>:CORRection:IMODulation:UTONe[:ACQuire]`).

Suffix:

<Ch> Calibrated channel number

<PhyPt> Port number. This suffix is ignored; the analyzer calibrates the port that is selected as a receive port for the intermodulation measurement (`[SENSe<Ch>:]FREQuency:IMODulation:RECeiver`).

Setting parameters:

<CalOnlyPortFreq> **OFF | 0**
The reference receiver at port <CalPort> is calibrated for all frequencies that are relevant for any of the ports.

ON | 1
Only those frequencies are calibrated that are relevant for this port. This results in shorter calibration times.

*RST: 0

Example: See `SOURce<Ch>:POWer<PhyPt>:CORRection:IMODulation:LTONe[:ACQuire]`

Usage: Setting only

Manual operation: See "Start Cal Sweep" on page 342

[SENSe<Ch>:]CORRection:POWer<PhyPt>[:STATe] <Boolean>

Enables or disables the receiver power calibration for channel `<Ch>` and for the received waves $b_{<PhyPt>}$. The setting command is disabled unless the received waves have been power calibrated (`[SENSe<Ch>:]CORRection:POWer<PhyPt>:ACQuire BWAVe, ...`). The query always returns a result.

Suffix:

<Ch> Calibrated channel number

<PhyPt> Calibrated port number

Parameters:

<Boolean> Enables (ON) or disables (OFF) the receiver power calibration for the received waves $b_{<PhyPt>}$.

*RST: OFF

Example: `*RST; CORR:POW:ACQ BWAVe,1,PORT,2`
Perform a receiver power calibration of the wave b1 using port 2 as a source port, assuming that the source power a_2 is correct. A through connection from port 2 to port 1 is needed.
`CORR:POW?`
Check whether the calibration is applied (the response is `1`).
`CALC:PAR:MEAS 'TRC1', 'B1D2'`
Select b_1 as a measured quantity for the default trace.
`CALC:MARK ON; MARK:Y?; :SOUR:POW?`
Create marker no. `<Mk>` in the center of the sweep range and query the measurement value. The calibrated power of the received wave b_1 is approx. equal to the default source power value.
See also example for `[SENSe<Ch>:]CORRection:POWer<PhyPt>:ACQuire`.

Manual operation: See "Port Overview" on page 657

[SENSe<Chn>:]CORRection:PSTate?

Gets the power calibration label of the active trace in channel <Chn>.

See Section 4.5.6.4, "Power calibration labels", on page 191 for the possible return values.

Suffix:
<Chn> Channel number used to identify the active trace

Usage: Query only

Manual operation: See "Apply" on page 598

[SENSe<Ch>:]CORRection:SMATrix:CDATa <ErrorTerm>, <SourceTestPort>, <LoadTestPort>, <SourceVNAPort>, <LoadVNAPort>, <CorrectionData>...
[SENSe<Ch>:]CORRection:SMATrix:CDATa:PORT<PhyPt> <ErrorTerm>, <SourceTestPort>, <LoadTestPort>, <SourceVNAPort>, <LoadVNAPort>, <CorrectionData>...

Writes or reads system error correction data in the presence of switching matrices.

Same command structure and logic as for `[SENSe<Ch>:]CORRection:CDATa...` commands, except for additional parameters `SourceVNAPort` and `LoadVNAPort` that restrict the returned correction data to the measurement path between by the respective physical VNA ports (if any). See Section 4.7.30.5, "Multiport calibration", on page 277 for details.

Suffix:
<Ch>

<PhyPt>

Parameters:
<ErrorTerm>

<SourceTestPort>

<LoadTestPort>

<SourceVNAPort> Number of the VNA port connected to the source test port (via switch matrix)

<LoadVNAPort> Number of the VNA port connected to the load test port (via switch matrix)

<CorrectionData> Correction data either in ASCII or block data format, depending on the current `FORMat[:DATA]` setting.

[SENSe<Ch>:]CORRection[:STATe] <Boolean>

Enables or disables the system error correction for channel `<Ch>`.

Suffix:
<Ch> Calibrated channel number

Parameters:
<Boolean> Enables (ON) or disables (OFF) the correction.
*RST: ON

Example:
```
*RST; :CORR?
```
Reset the instrument and query whether channel 1 is system error corrected. The response is `1`.

Manual operation: See "User Cal Active" on page 655

[SENSe<Ch>:]CORRection:STIMulus?
[SENSe<Ch>:]CORRection:STIMulus:PORT<PhyPt>?

Queries the stimulus values of the active calibration. A calibration must be selected before the command is executed; see example.

Suffix:
<Ch> Channel number of the calibrated channel

<PhyPt> Physical port number, used to select a specific frequency axis in arbitrary mode (with option R&S ZNB-K4).

Example:
```
*RST; :CORR:COLL:METH:DEF 'Test',RSHort,1
```
Select a one-port normalization at port 1 with a short standard as calibration type.
```
CORR:STIM?
```
Query the stimulus frequencies. The response contains 201 frequency values.
```
CORR:COLL:SEL SHOR,1
```
Measure a short standard connected to port 1 and store the measurement results of this standard.
```
CORR:COLL:SAVE:SEL
```
Calculate the system error correction data and apply it to the active channel.

Usage: Query only

Manual operation: See "Apply" on page 591

[SENSe<Chn>:]CORRection:SSTate?

Returns the system error correction state label of the active trace in channel <Chn>. The response is a string variable containing the calibration state label in the trace list ('CAL', 'CAI', 'CAL OFF' ..; see Section 4.5.4, "Calibration state labels", on page 175).

Suffix:
<Chn> Channel number used to identify the active trace

Example: See [SENSe<Ch>:]CORRection:DATE?

Usage: Query only

Manual operation: See "Apply" on page 591

7.3.14.9 [SENSe:]FREQuency...

The [SENSe:]... commands set frequency-related parameters, especially the measurement and display ranges for the different sweep types. The frequency ranges for the different instrument models are listed below; for more details refer to the data sheet.

Table 7-18: Frequency ranges of R&S ZNB analyzers

Frequency settings	Start, Stop	Center	Span
R&S ZNB4	9 kHz to 4.5 GHz	>9 kHz to <4.5 GHz	1 Hz to 4.499991 GHz
R&S ZNB8	9 kHz to 8.5 GHz	>9 kHz to <8.5 GHz	1 Hz to 8.499991 GHz
R&S ZNB20	100 kHz to 20 GHz	>100 kHz to <20 GHz	1 Hz to 19.9999 GHz
R&S ZNB	100 kHz to 26.5 GHz	>100 kHz to <26.5 GHz	1 Hz to 26.4999 GHz
R&S ZNB40 variant 72	10 MHz to 40 GHz	>10 MHz to <40 GHz	1 Hz to 39.99 GHz
R&S ZNB40 variants 82, 83 and 84	100 kHz to 40 GHz	>100 kHz to <40 GHz	1 Hz to 39.9999 GHz
R&S ZNB43	100 kHz to 43.5 GHz	>100 kHz to <43.5 GHz	1 Hz to 43.4999 GHz

Option R&S ZNB-B1, bias tees

If Bias tees are installed on a R&S ZNB4 or R&S ZNB8, the frequency range starts at 100 kHz.

[SENSe<Ch>:]FREQuency:CENTer 1398
[SENSe<Ch>:]FREQuency:CONVersion 1399
[SENSe<Ch>:]FREQuency:CONVersion:ARBitrary 1399
[SENSe<Ch>:]FREQuency:CONVersion:ARBitrary:PMETer<Pmtr> 1400
[SENSe<Ch>:]FREQuency:CONVersion:GAIN:LMCorrection 1401
[SENSe<Ch>:]FREQuency:CONVersion:MIXer:FIXed<Stg> 1401

[SENSe<Ch>:]FREQuency:CONVersion:MIXer:FUNDamental ... 1402
[SENSe<Ch>:]FREQuency:CONVersion:MIXer:IFPort ... 1402
[SENSe<Ch>:]FREQuency:CONVersion:MIXer:LOMultiplier<Stg> ... 1403
[SENSe<Ch>:]FREQuency:CONVersion:MIXer:LOPort<Stg> ... 1403
[SENSe<Ch>:]FREQuency:CONVersion:MIXer:MFFixed ... 1404
[SENSe<Ch>:]FREQuency:CONVersion:MIXer:RFMultiplier ... 1405
[SENSe<Ch>:]FREQuency:CONVersion:MIXer:RFPort ... 1406
[SENSe<Ch>:]FREQuency:CONVersion:MIXer:STAGes ... 1406
[SENSe<Ch>:]FREQuency:CONVersion:MIXer:TFRequency<Stg> ... 1406
[SENSe<Ch>:]FREQuency[:CW] ... 1407
[SENSe<Ch>:]FREQuency:FIXed ... 1407
[SENSe<Ch>:]FREQuency:IMODulation:CONVersion ... 1408
[SENSe<Ch>:]FREQuency:IMODulation:CWFixed ... 1408
[SENSe<Ch>:]FREQuency:IMODulation:LTONe ... 1408
[SENSe<Ch>:]FREQuency:IMODulation:ORDer<Im>[:STATe] ... 1409
[SENSe<Ch>:]FREQuency:IMODulation:RECeiver ... 1409
[SENSe<Ch>:]FREQuency:IMODulation:SPECtrum:MORDer ... 1410
[SENSe<Ch>:]FREQuency:IMODulation:SPECtrum[:STATe] ... 1410
[SENSe<Ch>:]FREQuency:IMODulation:TDIStance ... 1411
[SENSe<Ch>:]FREQuency:IMODulation:UTONe ... 1412
[SENSe<Ch>:]FREQuency:SBANd ... 1412
[SENSe<Ch>:]FREQuency:SEGMent:AXIS ... 1413
[SENSe<Ch>:]FREQuency:SPAN ... 1414
[SENSe<Ch>:]FREQuency:STARt ... 1414
[SENSe<Ch>:]FREQuency:STOP ... 1414

[SENSe<Ch>:]FREQuency:CENTer <CenterFreq>

Defines the center of the measurement and display range for a frequency sweep (sweep range). The default center frequency is the center of the analyzer's maximum frequency range: (f_{MIN} + f_{MAX})/2. The range depends on the instrument model; see Table 7-18.

Suffix:
<Ch> Channel number

Parameters:
<CenterFreq> Center frequency of the sweep

Increment: 0.1 kHz
Default unit: Hz

Example:
```
*RST; :SYST:FREQ? MIN; :SYST:FREQ? MAX
```
Query the frequency range of the analyzer.
```
FREQ:CENT 100MHz
```
Set center frequency to 100 MHz.
```
FREQ:SPAN 50000
```
Set frequency span to 50 kHz.

Manual operation: See "Start Frequency / Stop Frequency / Center Frequency / Span Frequency" on page 527

Note: The measurement range defined by means of the center frequency and the current span ([SENSe<Ch>:]FREQuency:SPAN) must not exceed the allowed frequency range of the analyzer. If necessary, the span is reduced to min (<CenterFreq> – f_{MIN}, f_{MAX}– <CenterFreq>).

[SENSe<Ch>:]FREQuency:CONVersion <ConversionMode>

Enables a frequency conversion measurement mode for channel <Ch>.

Suffix:
<Ch> Channel number

Parameters:
<ConversionMode> FUNDamental | ARBitrary | MIXer | IMODulation

FUNDamental
Measurement of the fundamental signal (non frequency-converting mode). This selection cancels all port-specific frequency and power settings (see example).

ARBitrary
Frequency conversion mode (arbitrary port frequencies). This mode is automatically activated when a port-specific frequency is defined.

MIXer
Scalar mixer mode, configured using [SENSe<Ch>:]FREQuency:CONVersion:MIXer... commands.

IMODulation
Intermodulation measurement mode, configured using the [SENSe<Ch>:]FREQuency:IMODulation... commands. Requires R&S ZNB-K14.

*RST: FUNDamental

Example: See SOURce<Ch>:FREQuency<PhyPt>:CONVersion:ARBitrary:IFRequency

Options: R&S ZNB-K4

Manual operation: See "Reset Freq Conv" on page 324

[SENSe<Ch>:]FREQuency:CONVersion:ARBitrary <Numerator>, <Denominator>, <Offset>, <SweepType>

Unused command.

The R&S ZNB doesn't allow to set a separate receiver frequency for Frequency conversion measurements. Use SOURce<Ch>:FREQuency<PhyPt>:CONVersion:ARBitrary:IFRequency to define the port-specific frequency (range or CW).

[SENSe<Ch>:]FREQuency:CONVersion:ARBitrary:PMETer<Pmtr> <BaseRaNum>, <BaseRatDen>, <Offset>, <SweepType>[, <OffsRatNum>[, <OffsRatDen>]]

Defines the receiver frequency of a power meter used for frequency-converting measurements. The receiver frequency is either a range (for frequency sweeps) or a CW frequency (for power, time and CW Mode sweeps). The receiver frequency is valid for all ports.

<Numerator>, <Denominator> and <Offset> are parameters of the frequency formula. The receiver frequency f_r is calculated according to

f_r = <BaseRatNum>/<BaseRatDen> * f_b + <OffsRatNum>/<OffsRatDen> * <Offset>,

where f_b represents the channel base frequency (parameter `SWEep`). For parameters `CW` or `FIXed`, f_b = 0.

Note:

- The default frequency or frequency range corresponds to the sweep range or CW frequency of the analyzer.
- The converted frequency or frequency range must be within the power meter's receiver frequency range. Nevertheless, the frequency formula is applied even if the analyzer returns an error message, because the frequency is outside the allowed range.

Suffix:

<Ch> Channel number

<Pmtr> Power meter number

Parameters:

<BaseRatNum>, <BaseRatDen>
Base ration numerator and denominator.
The specified numerator is rounded to the next higher integer; zero is not allowed. The specified denominator must be positive. It is rounded to the next higher integer.

<Offset>
Offset value, rounded (integer)

*RST: 0
Default unit: Hz

<SweepType>
CW | FIXed | SWEep

SWEep - the full formula is applied. For frequency sweeps, the command defines a sweep range.
CW | FIXed - the reduced formula with f_b = 0 is applied; the command defines a fixed frequency.

*RST: SWEep

<OffsRatNum>, <OffsRatDen>
Offset ratio numerator and denominator.
The specified numerator is rounded to the next higher integer; zero is not allowed. The specified denominator must be positive. It is rounded to the next higher integer.

Example: `*RST; FREQ:STAR 1E+9; STOP 1.1E+9`
Reset the analyzer (activating a frequency sweep) and set the sweep range between 1 GHz and 1.1 GHz.
`SYST:COMM:RDEV:PMET1:DEF 'USB Power Meter', 'NRP-Z55', 'usb', '100045'`
Configure an R&S NRP power meter as external power meter no. 1, assigning the name USB Power Meter and an serial number 100045.
`SENS:FREQ:CONV:ARB:PMET 2, 1, 1E+9, SWE`
Convert the receiver frequency to the range between 3 GHz and 3.2 GHz.

Manual operation: See "Frequency Conversion Formula" on page 689

[SENSe<Ch>:]FREQuency:CONVersion:GAIN:LMCorrection <Boolean>

Enables or disables load match correction for frequency transmission S-parameters (conversion gain factors in case of frequency-converting measurements).

Suffix:
<Ch> Channel number

Parameters:
<Boolean> *RST: ON

Example: See `SOURce<Ch>:FREQuency<PhyPt>:CONVersion:ARBitrary:IFRequency`

Manual operation: See "Load Match Correction" on page 656

[SENSe<Ch>:]FREQuency:CONVersion:MIXer:FIXed<Stg> <InputOutput>

Selects the mixer input or output signal which is at the fixed frequency defined via `[SENSe<Ch>:]FREQuency:CONVersion:MIXer:MFFixed`.

Suffix:
<Ch> Channel number.

<Stg> Counter for fixed signal (1 if only one mixer stage is used, 1 or 2 for 2 mixer stages).

Parameters:
<InputOutput> RF | LO | LO1 | LO2 | IF

RF
Mixer input signal

LO | LO1
Local oscillator signal no. 1

LO2
Local oscillator signal no. 2, for 2-stage mixer measurements (`[SENSe<Ch>:]FREQuency:CONVersion:MIXer:STAGes`)

IF
Mixer IF output signal (of the 2nd mixer, if avalable)
If mixer mode is IQ Mixer, both I and Q mixer IF output are selected.

Example: See `[SENSe<Ch>:]FREQuency:CONVersion:MIXer:MFFixed`

Options: R&S ZNB-K4

[SENSe<Ch>:]FREQuency:CONVersion:MIXer:FUNDamental <InputOutput>

Selects the mixer input or output signal which is at the fundamental frequency (channel frequency, to be defined by means of `[SENSe<Ch>:]FREQuency:STARt`, `[SENSe<Ch>:]FREQuency:STOP`, `[SENSe<Ch>:]FREQuency:CW` etc.).

Suffix:
<Ch> Channel number.

Parameters:
<InputOutput> RF | LO | LO1 | LO2 | IF | IF2

RF
Mixer input signal

LO | LO1
Local oscillator signal no. 1

LO2
Local oscillator signal no. 2, for 2-stage mixer measurements (`[SENSe<Ch>:]FREQuency:CONVersion:MIXer:STAGes`)

IF
IF – Mixer output signal (of the 2nd mixer, if avalable)

Example: See `[SENSe<Ch>:]FREQuency:CONVersion:MIXer:MFFixed`

Manual operation: See "Frequency tab" on page 328

[SENSe<Ch>:]FREQuency:CONVersion:MIXer:IFPort <PortNumber>

Selects an analyzer port as receive port for the IF signal.

Suffix:
<Ch> Channel number

Parameters:
<PortNumber> Range: 1 to number of test ports
*RST: 2

Example: See `[SENSe<Ch>:]FREQuency:CONVersion:MIXer:MFFixed`

Manual operation: See "Port selection" on page 327

[SENSe<Ch>:]FREQuency:CONVersion:MIXer:LOMultiplier<Stg> <Numerator>, <Denominator>

Selects the frequency conversion factors for the LO signal.

The conversion factor is a ratio of two positive integers: numerator and denominator.

Suffix:

<Ch> Channel number

<Stg> Mixer stage signals LO 1 or LO 2; see `[SENSe<Ch>:]FREQuency:CONVersion:MIXer:STAGes`

Parameters:

<Numerator>
Range: 1, 2, 3 ...
*RST: 1

<Denominator>
Range: 1, 2, 3 ...
*RST: 1

Example: See `[SENSe<Ch>:]FREQuency:CONVersion:MIXer:MFFixed`

Manual operation: See "Multiplication tab" on page 329

[SENSe<Ch>:]FREQuency:CONVersion:MIXer:LOPort<Stg> <SourceType>[, <SourceNumber>]

Selects an analyzer or an external generator port as a signal source for the LO1 or LO2 signal.

Note:

If External switch matrices are part of the RF connection configuration, operation with more than one internal source is *not* supported.

In this case only external generators can be used as local oscillators.

Suffix:

<Ch> Channel number

<Stg> Mixer stage (signals LO 1 or LO 2; see `[SENSe<Ch>:]FREQuency:CONVersion:MIXer:STAGes`).

Parameters:

<SourceType> PORT | GENerator | NONE

PORT – analyzer port
GENerator – external generator
NONE – LO signal is not controlled by the analyzer.

*RST: NONE

<SourceNumber> Number of analyzer port or generator.

Example: See `[SENSe<Ch>:]FREQuency:CONVersion:MIXer:MFFixed`

Manual operation: See "Port selection" on page 327

[SENSe<Ch>:]FREQuency:CONVersion:MIXer:MFFixed <InputOutput>[, <FixedFrequency>]

Assigns a fixed frequency to one of the input/output signals of a mixer setup. The fixed frequency setting becomes active if the port is selected as a port with fixed frequency (`[SENSe<Ch>:]FREQuency:CONVersion:MIXer:FIXed<Stg>`).

Suffix:

<Ch> Channel number

Parameters:

<InputOutput> RF | LO | LO1 | LO2 | IF

RF
Mixer input signal

LO | LO1
Local oscillator signal no. 1

LO2
Local oscillator signal no. 2, for 2-stage mixer measurements (`[SENSe<Ch>:]FREQuency:CONVersion:MIXer:STAGes`)

IF
Mixer IF output signal (of the 2nd mixer, if avalable)
If mixer mode is IQ Mixer, both I and Q mixer outputs are assigned the same fixed frequency.

<FixedFrequency> Default unit: Hz

Example: The following example requires an instrument with two internal sources. Moreover, an external generator (Gen 1) must be connected to the analyzer and configured.

```
*RST; FREQ:CONV:MIX:STAG 2
```

Select a mixer measurement with two mixer stages.

```
FREQ:CONV:MIX:RFPort 1; IFPort 2
```

Select the analyzer port 1 as a source port for the RF signal, port 2 as a receive port for the IF signal.

```
FREQ:CONV:MIX:LOPort1 PORT, 3; LOPort2 GEN, 1
```

Select port 3 as a source port for the local oscillator LO1, port 3 as a source for LO 2.

```
FREQ:CONV:MIX:RFMultiplier 2, 1; LOMultiplier1 2, 1 ; LOMultiplier2 2, 1
```

Define frequency conversion factors 2 for the RF signals and the LO signals.

```
FREQ:STARt 1GHz; STOP 2 GHZ
```

Select a sweep range between 1 GHz and 2 GHz.

```
FREQ:CONV:MIX:FIXed1 LO1; FIXed2 LO2
```

Assign fixed frequency settings to the two LO signals.

```
FREQ:CONV:MIX:FUNDamental RF; MFFixed LO1, 10 MHz; MFFixed LO2, 10 MHz
```

Assign the channel base frequency (sweep range) to the RF signal, fixed frequencies of 10 MHz to both LO signals.

```
FREQ:CONV:MIX:TFRequency1 DCUP; TFRequency2 UCON
```

Select down-conversion (USB) at the first mixer-stage, up-conversion (USB) at the second mixer stage.

```
SOUR:FREQ:CONV:MIX:PMODe RF, FUNDamental; PMODE LO1, FIXed; PMODe LO2, FIXed; PMODe IF, FUNDamental
```

Assign the channel base power to the RF and IF signals, a fixed power to the LO signals.

```
SOUR:FREQ:CONV:MIX:PMFixed LO1, -20; PMFixed LO2, -10
```

Define the fixed powers of the LO signals.

Options: R&S ZNB-K4

Manual operation: See "Frequency tab" on page 328

[SENSe<Ch>:]FREQuency:CONVersion:MIXer:RFMultiplier <Numerator>, <Denominator>

Selects the frequency conversion factor for the RF signal. The conversion factor is a ratio of two positive integers (numerator, denominator).

Suffix:

<Ch> Channel number

Parameters:

<Numerator>
Range: 1, 2, 3 ...
*RST: 1

<Denominator>
Range: 1, 2, 3 ...
*RST: 1

Example: See `[SENSe<Ch>:]FREQuency:CONVersion:MIXer:MFFixed`

Manual operation: See "Multiplication tab" on page 329

[SENSe<Ch>:]FREQuency:CONVersion:MIXer:RFPort <Port>

Selects a test port as source port for the RF signal.

Suffix:

<Ch> Channel number

Parameters:

<Port> Test port number
Range: 1 to number of test ports
*RST: 1

Example: See `[SENSe<Ch>:]FREQuency:CONVersion:MIXer:MFFixed`

Manual operation: See "Port selection" on page 327

[SENSe<Ch>:]FREQuency:CONVersion:MIXer:STAGes <Stages>

Selects the number of mixer stages for scalar mixer measurements.

Suffix:

<Ch> Channel number

Parameters:

<Stages> Number of mixer stages
Range: 1 | 2
*RST: 1

Example: See `[SENSe<Ch>:]FREQuency:CONVersion:MIXer:MFFixed`

Manual operation: See "2nd Mixer" on page 328

[SENSe<Ch>:]FREQuency:CONVersion:MIXer:TFRequency<Stg>
<IFConversionMode>

Selects the frequency conversion mode of the IF signal.

Suffix:

<Ch> Channel number.

<Stg> Mixer stage (1 or 2; see `[SENSe<Ch>:]FREQuency:CONVersion:MIXer:STAGes`.

Parameters:

<IFConversionMode> DCUPper | DCLower | UCONversion | BAND1 | BAND2

DCUPper | BAND2 – IF = RF - LO (down conversion upper sideband, equivalent to BAND2)
DCLower | BAND1– IF = LO - RF (down conversion lower sideband, equivalent to BAND1)
UCONversion – IF = LO + RF (up conversion)

*RST: DCUPper

Example:

```
*RST; FREQ:CONV:MIX:LOF 3 GHz
```

Reset the analyzer and specify a fixed frequency of 1 GHz, to be assigned to the LO signal.

```
FREQ:STAR 1 GHz; STOP 2 GHz
```

Define a sweep range between 3 and 4 GHz, to be assigned to the RF signal.

```
FREQ:CONV:MIX:TFR DCL
```

Select down conversion to the lower sideband, corresponding to an IF frequency range between 2 GHz and 1 GHz.

Manual operation: See "Conversion" on page 327

[SENSe<Ch>:]FREQuency[:CW] <FixedFreq>
[SENSe<Ch>:]FREQuency:FIXed <FixedFreq>

Defines the fixed (Continuous Wave, CW) frequency for all sweep types operating at fixed frequency ("Power", "Time", "CW Mode"). The two command forms `[SENSe<Ch>:]FREQuency[:CW]` and `[SENSe<Ch>:]FREQuency:FIXed` are equivalent.

This command also defines the center frequency of the intermodulation spectrum diagram (option R&S ZNB-K14). The frequency range depends on the R&S ZNB model; see Table 7-18.

Note: `[SENSe<Ch>:]FREQuency[:CW]|:FIXed` is equivalent to `SOURce<Ch>:FREQuency<PhyPt>[:CW]|:FIXed`. Source and receiver frequency are always equal; the four commands overwrite each other.

With option R&S ZNB-K4, Frequency Conversion Measurements, port-specific source and receiver frequencies can be defined; see `SOURce<Ch>:FREQuency<PhyPt>:CONVersion:ARBitrary:IFRequency`.

Suffix:

<Ch> Channel number

Parameters:

<FixedFreq> Fixed stimulus and analyzer frequency

*RST: 1 GHz
Default unit: Hz

Example: `FUNC "XTIMe:POW:A1"`
Activate a time sweep and select the wave quantity a_1 as measured parameter for channel and trace no. 1.
`FREQ:CW 100MHz`
Set the CW frequency to 100 MHz.

Manual operation: See "CW Frequency" on page 528

[SENSe<Ch>:]FREQuency:IMODulation:CONVersion <OFF>

Disables the intermodulation measurement and switches back to normal mode (no frequency conversion).

Suffix:
<Ch> Channel number

Setting parameters:
<OFF> OFF
Disable intermodulation measurement
*RST: n/a

Example: See `[SENSe<Ch>:]FREQuency:IMODulation:LTONe`

Usage: Setting only

Manual operation: See "Reset Freq Conv Intermod" on page 335

[SENSe<Ch>:]FREQuency:IMODulation:CWFixed <CWFixedToSettings>

Specifies the frequency which is kept constant during the sweep of the frequency difference between the upper and the lower tone. For intermodulation measurents.

Suffix:
<Ch> Channel number

Parameters:
<CWFixedToSettings> UPPertone | LOWertone | CENTer
Selects if the frequency of the upper tone or the lower tone is kept constant during the delta frequency sweep. Or if the center frequency between upper and lower tone is kept constant.

[SENSe<Ch>:]FREQuency:IMODulation:LTONe <SourceType>[, <SourceNumber>]

Selects the source for the lower tone signal that is used for the intermodulation measurement.

Suffix:
<Ch> Channel number

Parameters:
<SourceType> PORT
Lower tone source; internal source at port <arg1>

<SourceNumber> Number of the port for the internal source

Range: 1 to port number of the analyzer/number of external generators

*RST: n/a (for all parameters)

Example:

```
*RST; FREQ:IMOD:LTON PORT,1
```
Select port 1 as a source port for the lower tone.
```
FREQ:IMOD:UTON GEN,1
```
Select an external generator as a source for the upper tone (must be configured previously).
```
SOUR:POWer:CORR:IMOD:PORT 'COMBINER'
```
Select Extenal Device as a source port for the two tone signal.
```
FREQ:IMOD:REC 2
```
Select port 2 as the receiver port for the intermodulation measurement.
```
FREQ:IMOD:TDIS 2E6
```
Select a tone distance of 2 MHz.
```
FREQ:IMOD:ORD3 ON; :ORD5 ON; :ORD7 OFF; :ORD9 OFF
```
Enable the measurement of the intermodulation products up to the 5th order.
```
...
```
(Perform intermodulation measurement, evaluate results.)
```
FREQ:IMOD:CONV OFF
```
Disable intermodulation measurement, switch back to normal (non frequency-converting) mode.

Manual operation: See "Lower Tone" on page 339

[SENSe<Ch>:]FREQuency:IMODulation:ORDer<Im>[:STATe] <Boolean>

Enables or disables the measurement of the intermodulation products of order <IM order>.

Suffix:

<Ch> Channel number

<Im> Order of IM products

Parameters:

<Boolean> ON | OFF – enables or disables measurement

*RST: ON (for <Im> = 3), OFF (for <Im> > 3)

Example: See `[SENSe<Ch>:]FREQuency:IMODulation:LTONe`

Manual operation: See "Prepare Measurement of IM Order" on page 340

[SENSe<Ch>:]FREQuency:IMODulation:RECeiver <PortNumber>

Selects the receiving port for the intermodulation measurement.

Suffix:
<Ch> Channel number

Parameters:
<PortNumber> Analyzer port number
Range: 1 to the number of analyzer ports
*RST: 2

Example: See `[SENSe<Ch>:]FREQuency:IMODulation:LTONe`

Manual operation: See "Receiving Port" on page 339

[SENSe<Ch>:]FREQuency:IMODulation:SPECtrum:MORDer <IMOrder>

Defines the maximum order of intermodulation products for the intermodulation spectrum measurement.

Suffix:
<Ch> Channel number

Parameters:
<IMOrder> Maximum order of IM products
Range: 3 | 5 | 7 | 9
*RST: 3

Example: See `[SENSe<Ch>:]FREQuency:IMODulation:SPECtrum[:STATe]`

Manual operation: See "CW Mode Spectrum" on page 335

[SENSe<Ch>:]FREQuency:IMODulation:SPECtrum[:STATe] <Boolean>

Enables or disables the measurement of the intermodulation spectrum. If a new channel is desired, the trace must be created separately; see second example below.

Suffix:
<Ch> Channel number

Parameters:
<Boolean> ON | OFF - enables or disables intermodulation spectrum measurement.
*RST: OFF

Example: Intermodulation spectrum measurement within the active channel.

```
*RST; FREQ:IMOD:LTON PORT,1; UTON GEN, 1
```

Reset the instrument to create the default channel no. 1. Select port 1 as a source port for the lower tone, a (previously configured) external generator no. 1 as a source port for the upper tone.

```
FREQ:STAR 1GHZ; STOP 2GHz
```

Define a suitable sweep range to ensure that the analyzer can measure all intermodulation products.

```
SENS1:FREQ:IMOD:SPEC ON
```

Enable the intermodulation spectrum measurement for channel no. 1.

```
CALC1:PAR:CAT?
```

Query the traces in channel no. 1. The response is `'Trc1,B2D1SAM'`.

```
SENS1:FREQ 1.5 GHz; :SENS1:FREQ:IMOD:SPEC:MORD 9
```

Adjust the center frequency and the maximum IM order of the spectrum intermodulation diagram.

Example: Intermodulation spectrum measurement in a new channel.

```
*RST; FREQ:IMOD:LTON PORT,1; UTON GEN, 1
```

Reset the instrument to create the default channel no. 1. Select port 1 as a source port for the lower tone, a (previously configured) external generator no. 1 as a source port for the upper tone.

```
FREQ:STAR 1GHZ; STOP 2GHz
```

Define a suitable sweep range to ensure that the analyzer can measure all intermodulation products.

```
CALC2:PAR:SDEF 'Trc2', 'S21'
```

Create a new channel no. 2 and a trace named 'Trc2'. Select S_{21} as a measured quantity.

```
DISP:WIND:TRAC2:FEED 'Trc2'
```

Display the new trace in diagram no. 1.

```
SENS2:FREQ:IMOD:SPEC ON
```

Enable the intermodulation spectrum measurement for the new channel no. 2.

```
CALC2:PAR:CAT?
```

Query the traces in channel no. 2. The response is `'Trc2,B2D1SAM'`.

Manual operation: See "CW Mode Spectrum" on page 335

[SENSe<Ch>:]FREQuency:IMODulation:TDIStance <ToneDistance>

Defines the tone distance (frequency offset) between the upper and the lower tone.

Suffix:

<Ch> Channel number

Parameters:

<ToneDistance> Upper tone frequency minus lower tone frequency.

Range: Lower limit: 0 Hz, upper limit depending on the instrument model and the sweep range for the lower tone.
*RST: 1 MHz
Default unit: Hz

Example: See `[SENSe<Ch>:]FREQuency:IMODulation:LTONe`

Manual operation: See "Tone Distance" on page 340

[SENSe<Ch>:]FREQuency:IMODulation:UTONe <SourceType>[, <arg1>]

Selects the source for the upper tone signal that is used for the intermodulation measurement.

Note:

If External switch matrices are part of the RF connection configuration, operation with more than one internal source is *not* supported. In this case only external generators can be used as local oscillators.

Suffix:

<Ch> Channel number

Parameters:

<SourceType> PORT | GENerator | NONE

Upper tone source:
NONE – no source selected (for query only)
PORT – internal second source at port <arg1>
GENerator – configured external generator no. <arg1>

*RST: NONE

<arg1> Analyzer port number or generator number

Range: 1 to number of ports of the analyzer or number of configured external generators
*RST: n/a (no analyzer or generator port selected)

Example: See `[SENSe<Ch>:]FREQuency:IMODulation:LTONe`

Manual operation: See "Upper Tone" on page 339

[SENSe<Ch>:]FREQuency:SBANd <Sideband>

Defines whether the analyzer measures with a local oscillator frequency LO below or above the RF input frequency.

Tip: In a segmented frequency sweep, it is possible to set the sideband (`SBANd`) parameter individually for each segment; see `[SENSe<Ch>:]SEGMent<Seg>:DEFine`. The `[SENSe<Ch>:]FREQuency:SBANd` setting is global and not valid for segmented sweeps. The two sideband settings do not overwrite each other.

Suffix:
<Ch> Channel number. This suffix is ignored; the setting applies to all channels in the active recall set.

Parameters:
<Sideband> POSitive | NEGative | AUTO

POSitive
LO > RF; the LO frequency is always above the measured RF frequency.

NEGative
LO < RF; the LO frequency is always below the measured RF frequency.

AUTO
The analyzer auto-selects the LO frequency, depending on the receiver (RF) frequency.

*RST: AUTO

Example: `*RST; :SWE:TYPE?; :FREQ:SBAN?`
Query the *RST values for the sweep type and the sideband setting. The response is `LIN` (linear frequency sweep) and `AUTO` (automatic setting of the LO frequency).

Manual operation: See "Image Suppr." on page 699

[SENSe<Ch>:]FREQuency:SEGMent:AXIS <Scale>

Selects either frequency based or point based x-axis for segmented sweeps.

Suffix:
<Ch> Channel number

Parameters:
<Scale> POINt | FREQuency

Example:
```
SENSe:SEGMent:INSert 1MHZ, 1.1MHZ, 101, -21DBM, 0.5S, 0, 10KHZ
SENSe:SEGMent:INSert 2MHZ, 3MHZ, 101, -21DBM, 0.5S, 0, 10KHZ
```
Create two sweep segments with different frequency spans, each with 101 sweep points.
```
SENSe:SEGMent:FREQuency:AXIS POINt
```
Select the point based frequency axis. The first 101 sweep points are distributed over the left half of the diagram, the second 101 points over the right half.

Manual operation: See "Seg X-Axis" on page 544

[SENSe<Ch>:]FREQuency:SPAN <Span>

Defines the width (span) of the measurement and display range for a frequency sweep (sweep range). The default span equals to the maximum frequency range of the analyzer: $f_{MAX} - f_{MIN}$.

The range depends on the instrument model; see Table 7-18Table 7-18.

Suffix:
<Ch> Channel number

Parameters:
<Span> Frequency span of the sweep

Increment: 0.1 kHz
Default unit: Hz

Example: See `[SENSe<Ch>:]FREQuency:CENTer`

Manual operation: See "Start Frequency / Stop Frequency / Center Frequency / Span Frequency" on page 527

Note: The measurement range defined by means of the span and the current center frequency (`[SENSe<Ch>:]FREQuency:CENTer`), must not exceed the allowed frequency range of the analyzer. If necessary, the center frequency is adjusted to f_{MIN} + <Span>/2 or f_{MAX} - <Span>/2.

[SENSe<Ch>:]FREQuency:STARt <FreqVal>
[SENSe<Ch>:]FREQuency:STOP <FreqVal>

These commands defines the start and stop frequency for a frequency sweep. The values also define the display range in a Cartesian diagram. The default start and stop frequencies equal to the minimum and maximum frequency of the analyzer.

The ranges depend on the instrument model; see Table 7-18.

Suffix:
<Ch> Channel number

Parameters:
<FreqVal> Start and stop frequency of the sweep

Increment: 0.1 kHz
Default unit: Hz

Example:
```
*RST; FREQ:STAR 10MHz
```
Activate a frequency sweep and set the start frequency to 10 MHz.
```
FREQ:STOP 100MHz
```
Set the stop frequency to 100 MHz.

Manual operation: See "Start Frequency / Stop Frequency / Power CW Frequency" on page 341

Note: If the start frequency entered is greater than the current stop frequency, the stop frequency is set to the start frequency plus the minimum frequency span (`[SENSe<Ch>:]FREQuency:SPAN`).

If the stop frequency entered is smaller than the current start frequency, the start frequency is set to the stop frequency minus the minimum frequency span.

7.3.14.10 [SENSe:]HARMonic...

These commands Implement functions related to harmonic grids for time domain transformation.

[SENSe<Ch>:]HARMonic?....1415
[SENSe<Ch>:]HARMonic:AUTO....1415
[SENSe<Ch>:]HARMonic:DLENgth:DATA....1415
[SENSe<Ch>:]HARMonic:ELENgth:DATA....1416
[SENSe<Ch>:]HARMonic:MLENgth:DATA....1416
[SENSe<Ch>:]HARMonic:PERMittivity:DATA....1416
[SENSe<Ch>:]HARMonic:RTIMe:DATA....1417
[SENSe<Ch>:]HARMonic:RTIMe:THReshold....1417
[SENSe<Ch>:]HARMonic:VELocity:DATA....1417

[SENSe<Ch>:]HARMonic?

Queries whether the current frequency grid is harmonic.

Suffix:
<Ch> Channel number

Return values:
<arg0>

Usage: Query only

Manual operation: See "Is the Current Grid Harmonic?" on page 363

[SENSe<Ch>:]HARMonic:AUTO <arg0>

Turns the "Automatic Harmonic Grid" function ON or OFF.

Suffix:
<Ch> Channel number

Parameters:
<arg0> *RST: OFF (0)

Options: R&S ZNB-K2

Manual operation: See "Automatic Harmonic Grid" on page 363

[SENSe<Ch>:]HARMonic:DLENgth:DATA <DUTLength>

Sets/gets the expected maximum time delay through the DUT.

The set command automatically activates the "Automatic Harmonic Grid" function, i.e. it sets `[SENSe<Ch>:]HARMonic:AUTO` to ON.

Suffix:
<Ch> Channel number

Parameters:
<DUTLength> Default unit: s

Options: R&S ZNB-K20

Manual operation: See "Max. DUT Measure Delay / Max. El. Length / Max. Mech. Length" on page 379

[SENSe<Ch>:]HARMonic:ELENgth:DATA <DUTELength>

Sets/gets the expected maximum electrical length of the DUT.

The set command automatically activates the "Automatic Harmonic Grid" function, i.e. it sets `[SENSe<Ch>:]HARMonic:AUTO` to ON.

Suffix:
<Ch> Channel number

Parameters:
<DUTELength> Default unit: m

Options: R&S ZNB-K20

Manual operation: See "Max. DUT Measure Delay / Max. El. Length / Max. Mech. Length" on page 379

[SENSe<Ch>:]HARMonic:MLENgth:DATA <DUTMLength>

Sets/gets the expected maximum mechanical length of the DUT.

The set command automatically activates the "Automatic Harmonic Grid" function, i.e. it sets `[SENSe<Ch>:]HARMonic:AUTO` to ON.

Suffix:
<Ch> Channel number

Parameters:
<DUTMLength> Default unit: m

Options: R&S ZNB-K20

Manual operation: See "Max. DUT Measure Delay / Max. El. Length / Max. Mech. Length" on page 379

[SENSe<Ch>:]HARMonic:PERMittivity:DATA <DUTPermittivity>

Sets/gets the (relative) permittivity of the DUT.

The set command automatically activates the "Automatic Harmonic Grid" function, i.e. it sets `[SENSe<Ch>:]HARMonic:AUTO` to ON.

Suffix:
<Ch> Channel number

Parameters:
<DUTPermittivity>

Options: R&S ZNB-K20

Manual operation: See "Permittivity / Velocity Factor" on page 379

[SENSe<Ch>:]HARMonic:RTIMe:DATA <DUTRiseTime>

Sets/gets the minimum rise time the user wishes to measure on the DUT.

Use `[SENSe<Ch>:]HARMonic:RTIMe:THReshold` to select the underlying rise time definition.

The set command automatically activates the "Automatic Harmonic Grid" function, i.e. it sets `[SENSe<Ch>:]HARMonic:AUTO` to ON.

Suffix:
<Ch> Channel number

Parameters:
<DUTRiseTime> Default unit: s

Options: R&S ZNB-K2

Manual operation: See "Rise Time" on page 380

[SENSe<Ch>:]HARMonic:RTIMe:THReshold <DUTRiseThreshold>

Defines how the rise time that is set using `[SENSe<Ch>:]HARMonic:RTIMe:DATA` shall be interpreted. Allows to select between rise time definitions 20%-80% (default) and 10%-90%.

The set command automatically activates the "Automatic Harmonic Grid" function, i.e. it sets `[SENSe<Ch>:]HARMonic:AUTO` to ON.

Suffix:
<Ch> Channel number

Parameters:
<DUTRiseThreshold> T1_9 | T2_8

Rise time definition:
T1_9: 10%-90%
T2_8: 20%-80%

*RST: T2_8

Options: R&S ZNB-K2

Manual operation: See "Rise Time Definition" on page 380

[SENSe<Ch>:]HARMonic:VELocity:DATA <DUTVelocity>

Sets/gets the velocity factor of the DUT.

The set command automatically activates the "Automatic Harmonic Grid" function, i.e. it sets `[SENSe<Ch>:]HARMonic:AUTO` to ON.

Suffix:
<Ch> Channel number

Parameters:
<DUTVelocity>

Options: R&S ZNB-K20

Manual operation: See "Permittivity / Velocity Factor" on page 379

7.3.14.11 [SENSe:]LPORt...

The `[SENSe:]LPORt...` commands define the reference impedances of the balanced ports.

[SENSe<Ch>:]LPORt<LogPt>:ZCOMmon 1418
[SENSe<Ch>:]LPORt<LogPt>:ZDIFferent 1418
[SENSe<Ch>:]LPORt<LogPt>:ZDEFault[:STATe] 1419

[SENSe<Ch>:]LPORt<LogPt>:ZCOMmon <RealPart>[, <ImaginaryPart>]
[SENSe<Ch>:]LPORt<LogPt>:ZDIFferent <RealPart>[, <ImaginaryPart>]

These commands specify the complex common mode and differential mode reference impedances for the balanced (logical) port numbered `<LogPt>`.

Use `[SENSe<Ch>:]LPORt<LogPt>:ZDEFault[:STATe]` to toggle between configured and default reference impedances.

Suffix:
<Ch> Channel number

<LogPt> Logical port number. The logical ports must be defined using `SOURce<Ch>:LPORt<LogPt> <PhysicalPort1>, <PhysicalPort2>`. An n port analyzer supports a maximum of n/2 (n even) or (n - 1)/2 (n odd) logical ports.

Parameters:
<RealPart> Real part of the port impedance.

Range: 1 mΩ to 10 MΩ
*RST: ZCOMmon: 25 Ω; ZDIFferent: 100 Ω (real impedances)
Default unit: Ohm

<ImaginaryPart> Imaginary part of the port impedance; may be omitted to define a real impedance.

Range: - 10 MΩ to 10 MΩ
*RST: 0 Ω (real impedances)
Default unit: Ohm

Example:
```
*RST; :CALC:PAR:DEL 'TRC1'
```
Reset the analyzer and delete the (default) trace for channel no. 1.
```
SOUR:LPOR1 1,2
```
Combine the physical ports no. 1 and 2 to define the balanced (logical) port no. 1.
```
LPOR1:ZDIF 27, 2
```
Specify a complex differential mode reference impedance of 27Ω + j * 2Ω for the defined port.

Manual operation: See "Single Ended Mode / Common Mode / Differential Mode" on page 302

[SENSe<Ch>:]LPORt<LogPt>:ZDEFault[:STATe] <arg0>

Allows to toggle between default and renormalized reference impedance(s) for logical port <LogPt>.

Suffix:
<Ch> Channel number

<LogPt> Logical port number

Parameters:
<arg0> ON (1): Use default impedance(s)
OFF (0): Use the redefined impedances defined via `[SENSe<Ch>:]PORT<PhyPt>:ZREFerence` for single-ended or via `[SENSe<Ch>:]LPORt<LogPt>:ZCOMmon` and `[SENSe<Ch>:]LPORt<LogPt>:ZDIFferent` for balanced logical ports

Manual operation: See "Use Default" on page 302

7.3.14.12 [SENSE:]NFIGure...

The `[SENSE:]NFIGure...` commands define properties of a Noise figure measurement.

[SENSe<Ch>:]NFIGure:CALibration:GTIMe ... 1420
[SENSe<Ch>:]NFIGure:CALibration:NTIMe ... 1420
[SENSe<Ch>:]NFIGure:DEFine ... 1420
[SENSe<Ch>:]NFIGure:GTIMe ... 1421
[SENSe<Ch>:]NFIGure:HARMonic[:MAXFrequency] ... 1421
[SENSe<Ch>:]NFIGure:HARMonic[:MINFrequency] ... 1421
[SENSe<Ch>:]NFIGure:HARMonic[:MAXimumorder] ... 1421
[SENSe<Ch>:]NFIGure:HARMonic[:STATe] ... 1422
[SENSe<Ch>:]NFIGure:IFConst ... 1422
[SENSe<Ch>:]NFIGure:IFConst:STATe ... 1422
[SENSe<Ch>:]NFIGure:NTIMe ... 1423
[SENSe<Ch>:]NFIGure:TEATtenuator ... 1423
[SENSe<Ch>:]NFIGure:VIRejection[:STATe] ... 1423

[SENSe<Ch>:]NFIGure:CALibration:GTIMe <DetectorTime>

Defines the detector time of the AVG detector which is used for the gain measurement during a NF calibration in channel <Ch>.

Suffix:
<Ch> Channel number

Parameters:
<DetectorTime> Detector time for the AVG detector
Default unit: s

Example: `SENSe1:NFIGure:CALibration:GTIMe 10ms`

Options: R&S ZNB-K30

Manual operation: See "Gain Det Meas Time" on page 355

[SENSe<Ch>:]NFIGure:CALibration:NTIMe <DetectorTime>

Defines the detector time of the RMS detector which is used for the noise measurement during a NF calibration in channel <Ch>.

Suffix:
<Ch> Channel number

Parameters:
<DetectorTime> Detector time for the RMS detector
Default unit: s

Example: `SENSe1:NFIGure:CALibration:NTIMe 50ms`

Options: R&S ZNB-K30

Manual operation: See "Noise Det Meas Time" on page 355

[SENSe<Ch>:]NFIGure:DEFine <receiving test port>, <driving test port>[, <further port pairs>...]
[SENSe<Ch>:]NFIGure:DEFine?

The setting initiates a noise figure measurement in channel <Ch> and defines the receive/drive test port pairs to be measured. The query returns the defined receive/drive test port pairs, or an empty string, if no pairs are defined.

Suffix:
<Ch> Channel number

Parameters:
<ReceivePort> Receiving test port number

<DrivePort> Driving test port number

Setting parameters:
<further port pairs> Additional receive/drive port pairs (comma-separated)

Example: `SENSe1:NFIGure:DEFine 2,1`
Initiates a noise figure measurement in channel 1, with drive port 1 and receive port 2.

Options: R&S ZNB-K30

Manual operation: See "New NF Channel/Convert Ch<n> to NF Channel" on page 349

[SENSe<Ch>:]NFIGure:GTIMe <DetectorTime>

Defines the detector time for the gain measurement in a noise figure channel <Ch>.

Suffix:
<Ch> Channel number

Parameters:
<DetectorTime> Detector time for the average detector
Default unit: s

Example: `SENSe1:NFIGure:GTIMe 10ms`

Options: R&S ZNB-K30

Manual operation: See "Gain Det Time" on page 352

[SENSe<Ch>:]NFIGure:HARMonic[:MAXFrequency] <FrequencyLimit>
[SENSe<Ch>:]NFIGure:HARMonic[:MINFrequency] <FrequencyLimit>

Sets the lower|upper frequency limit to which the sideband correction is used in a noise figure measurement in channel <Ch>.

Suffix:
<Ch> Channel number

Parameters:
<FrequencyLimit> Frequency limit at which the noise figure sideband correction is used.
Range: 10 MHz | 9 GHz
Default unit: Hz

Options: R&S ZNB-K30

Manual operation: See "Sideband Correction" on page 353

[SENSe<Ch>:]NFIGure:HARMonic[:MAXimumorder] <MaximumOrder>

Sets the sideband order of the sideband correction for a noise figure measurement in channel <Ch>.

Suffix:
<Ch> Channel number

Parameters:

<MaximumOrder> Sets the number of higher-order sidebands which are measured and corrected during calibration and measurement.

*RST: 3

Options: R&S ZNB-K30

Manual operation: See "Sideband Correction" on page 353

[SENSe<Ch>:]NFIGure:HARMonic[:STATe] <State>

Activates or deactivates the sideband correction for a noise figure measurement in channel <Ch>.

Suffix:

<Ch> Channel number

Parameters:

<State> Enables (1) or disables (0) the sideband correction of the noise figure calibration and measurement.

*RST: 1

Options: R&S ZNB-K30

Manual operation: See "Sideband Correction" on page 353

[SENSe<Ch>:]NFIGure:IFConst <IFFreq>

If manual selection is activated (`[SENSe<Ch>:]NFIGure:IFConst:STATe ON`), this command allows you to select an IF for noise figure measurements in channel <Ch>.

Suffix:

<Ch> Channel number

Parameters:

<IFFreq> Default unit: Hz

Options: R&S ZNB-K30

Manual operation: See "IF Frequency" on page 353

[SENSe<Ch>:]NFIGure:IFConst:STATe <Boolean>

Allows you to switch between automatic and manual IF selection for noise figure measurements in channel <Ch>

Suffix:

<Ch> Channel number

Parameters:

<Boolean> **OFF (0)**
Automatic selection

ON (1)
Manual selection using `[SENSe<Ch>:]NFIGure:IFConst`

Options: R&S ZNB-K30

Manual operation: See "IF Frequency" on page 353

[SENSe<Ch>:]NFIGure:NTIMe <DetectorTime>

Defines the detector time of the RMS detector which is used for the actual noise measurement during a NF measurement in channel <Ch>

Suffix:
<Ch> Channel number

Parameters:
<DetectorTime> Detector time for the RMS detector
Default unit: s

Example: `SENSe1:NFIGure:NTIMe 50ms`

Options: R&S ZNB-K30

Manual operation: See "Noise Det Time" on page 352

[SENSe<Ch>:]NFIGure:TEATtenuator <Temperature>

Sets the ambient temperature used for proper determination of the DUT input source noise during a noise figure measurement in channel <Ch>.

Suffix:
<Ch> Channel number

Parameters:
<Temperature> Ambient temperature in Kelvin.

Example: `SENSe1:NFIGure:TEATtenuator 320`

Options: R&S ZNB-K30

Manual operation: See "Amb Temp in Kelvin" on page 354

[SENSe<Ch>:]NFIGure:VIRejection[:STATe] <Boolean>

Defines whether virtual image rejection is used in noise figure channel <Ch>.

Suffix:
<Ch> Channel number

Parameters:
<Boolean> **ON (1)**
Virtual image rejection activated
OFF (0)
Virtual image rejection deactivated

Options: R&S ZNB-K30

Manual operation: See "Vir. Img. Rejection" on page 354

7.3.14.13 [SENSe:]PAE...

The `[SENSe:]PAE...` commands configure the measurement of the Power Added Efficiency of an active 2-port device.

[SENSe<Ch>:]PAE:DCINput:MAIN 1424
[SENSe<Ch>:]PAE:DCINput:SECondary 1424
[SENSe<Ch>:]PAE:PARameters:I 1424
[SENSe<Ch>:]PAE:PARameters:R 1425
[SENSe<Ch>:]PAE:PARameters:U 1425
[SENSe<Ch>:]PAE:TYPE 1425

[SENSe<Ch>:]PAE:DCINput:MAIN <Input>

Selects the first (main) DC INPUT connector for the PAE measurement (used for all measurement types; see example).

Suffix:
<Ch> Channel number

Parameters:
<Input> DC1 | DC2 | DC3 | DC4
Rear panel connectors DC INPUT 1 ... 4
*RST: DC1

Example: See `[SENSe<Ch>:]PAE:TYPE`

Manual operation: See "$U_m/U_{m1}/U_{m2}$/" on page 322

[SENSe<Ch>:]PAE:DCINput:SECondary <Input>

Selects the secondary DC INPUT connector for the PAE measurement (used for measurement type "Voltage and Current" only; see example). The main DC input (`[SENSe<Ch>:]PAE:DCINput:MAIN`) and the secondary DC input must be different from each other.

Suffix:
<Ch> Channel number

Parameters:
<Input> DC1 | DC2 | DC3 | DC4
Rear panel connectors DC INPUT 1 ... 4
*RST: DC2

Example: See `[SENSe<Ch>:]PAE:TYPE`

Manual operation: See "$U_m/U_{m1}/U_{m2}$/" on page 322

[SENSe<Ch>:]PAE:PARameters:I <Current>

Defines the constant DC supply current I_0 for the PAE measurement (measurement type: "Constant Current Source"; see example).

Suffix:
<Ch> Channel number

Parameters:
<Current> *RST: 0 A
Default unit: A

Example: See `[SENSe<Ch>:]PAE:TYPE`

Manual operation: See "Formula" on page 322

[SENSe<Ch>:]PAE:PARameters:R <Resistance>

Defines the resistance R of the precision resistor used for the PAE measurement (measurement types: "Constant Voltage Source" or "Voltage and Current"; see example).

Suffix:
<Ch> Channel number

Parameters:
<Resistance> Range: 0 Ohm to 1E+9 Ohm
*RST: 50 Ohm
Default unit: Ohm

Example: See `[SENSe<Ch>:]PAE:TYPE`

Manual operation: See "Formula" on page 322

[SENSe<Ch>:]PAE:PARameters:U <Voltage>

Defines the constant DC supply voltage U_0 for the PAE measurement (measurement type "Constant Voltage Source"; see example).

Suffix:
<Ch> Channel number

Parameters:
<Voltage> *RST: 0 V
Default unit: V

Example: See `[SENSe<Ch>:]PAE:TYPE`

Manual operation: See "Formula" on page 322

[SENSe<Ch>:]PAE:TYPE <Type>

Selects the measurement type for the Power Added Efficiency (PAE) measurement. The three measurement types involve different test setups, input parameters, and DC voltage measurements.

Suffix:
<Ch> Channel number

Parameters:

<Type> VOLTage | CURRent | VCURrent

Constant voltage source, constant current source, voltage and current

*RST: VOLTage

Example:

```
SENSe1:PAE:TYPE VOLTage
```
Select a measurement type with constant voltage source.
```
SENSe1:PAE:PARameters:U 5V; R 50Ohm
```
Define the parameters for the PAE measurement.
```
SENSe1:PAE:DCINput:MAIN DC1; :DC1:RANGe 20
```
Define the DC connector and power range

Alternative measurement type: constant current source.
```
SENSe1:PAE:TYPE CURRent
SENSe1:PAE:PARameters:I 100mA
SENSe1:PAE:DCINput:MAIN DC1; :DC1:RANGe 20
```

Alternative measurement type: voltange and current.
```
SENSe1:PAE:TYPE VCURrent
SENSe1:PAE:PARameters:R 50Ohm
SENSe1:PAE:DCINput:MAIN DC1; :DC1:RANGe 3
SENSe1:PAE:DCINput:SECondary DC2; :DC2:RANGe 20
```

For all measurement types:
```
CALCulate1:PARameter:SDEFine 'Trc1', 'PAE21'
DISPlay:WINDow1:TRACe:FEED 'Trc1'
```
Select a PAE measurement with port 1 as a source port, port 2 as a receive port.

Manual operation: See "Measurement Type" on page 321

7.3.14.14 [SENSe:]POWer...

The `[SENSe:]POWer...` commands configure the optional receiver step attenuators (see Section 4.7.24, "Receiver step attenuators", on page 264) and the AGC of the R&S ZNB.

See also `[SENSe<Ch>:]SEGMent<Seg>:POWer:GAINcontrol` on page 1445.

[SENSe<Ch>:]POWer:AGCMode:ACQuire 1427
[SENSe<Ch>:]POWer:AGCMode<PhyPt>:MEASure 1427
[SENSe<Ch>:]POWer:IFGain<PhyPt>:MEASure 1427
[SENSe<Ch>:]POWer:AGCMode:SAVE 1428
[SENSe<Ch>:]POWer:ATTenuation 1428
[SENSe<Ch>:]POWer:GAINcontrol 1429
[SENSe<Ch>:]POWer:GAINcontrol:ALL 1429
[SENSe<Ch>:]POWer:GAINcontrol:GLOBal 1430
[SENSe<Ch>:]POWer:LOSuppress[:STATe] 1431

[SENSe<Ch>:]POWer:AGCMode:ACQuire

Starts an AGC learn sweep.

Suffix:
<Ch> Channel number

Example: Before you start, make sure channel 1 is fully configured (ports, traces, ...).
`*RST; :SENSe1:POWer:GAINcontrol:GLOBal MANual`
Activate independent AGC settings for each drive port.
`SENSe1:POWer:AGCMode:ACQuire;*WAI`
Perform a Learn Sweep and wait until it has completed (asynchronous command)
`SENSe1:POWer:AGCMode:SAVE`
Apply the acquired AGC settings to channel 1.

Usage: Event

Manual operation: See "All Segments – Learn Sweep" on page 702

[SENSe<Ch>:]POWer:AGCMode<PhyPt>:MEASure <Mode>
[SENSe<Ch>:]POWer:IFGain<PhyPt>:MEASure <Mode>

These two equivalent commands activate manual gain control and select the IF gain in the measurement channels (b-waves). The IF gain in the reference channel (a-waves) is set automatically if not otherwise defined using `[SENSe<Ch>:]POWer:GAINcontrol`.

Note:

These commands are replaced by `[SENSe<Ch>:]POWer:GAINcontrol` and `[SENSe<Ch>:]POWer:GAINcontrol:GLOBal`. They are supported for compatibility with FW versions < 1.5.

Suffix:
<Ch> Channel number

<PhyPt> Test port number of the analyzer

Parameters:
<Mode> AUTO | LDIStortion | LNOise

AUTO
automatic/adaptive gain control (AGC) according to the RF input level

LDIStortion
fixed, small IF gain, for high input levels

LNOise
fixed, large IF gain, for low input levels

*RST: LDIS

Example: `*RST; :SENSe:POWer:IFG2:MEAS LNO; :SENSe:POWer:IFG1:MEAS?`
Optimize the received waves (measurement channel, b-waves) at port 2 for small input levels and query the AGC settings for the received waves at port 1. The response is `LDIS`.

[SENSe<Ch>:]POWer:AGCMode:SAVE

Apply the static AGC (Automatic Gain Control) settings obtained during an AGC Learning Sweep.

Suffix:
<Ch> Channel number

Example: see `[SENSe<Ch>:]POWer:AGCMode:ACQuire`

Usage: Event

Manual operation: See "All Segments – Learn Sweep" on page 702

[SENSe<Ch>:]POWer:ATTenuation <PortNo>[, <Att>]

Sets an attenuation factor for the received waves. This command is available if at least one of the Receiver step attenuators is installed.

For redefined physical ports (see `[SENSe:]UDSParams<Pt>:PARam`), the respective measurement receiver (b-wave) is significant. E.g. if physical port 1 is equipped with a step attenuator, then an attenuation factor can be applied to the (redefined) port receiving b1.

Note that in presence of External switch matrices, **all** VNA ports have to be equipped with receiver step attenuator option.

Suffix:
<Ch> Channel number

Parameters:
<PortNo> Physical port number
Instead of port numbers 1, 2, 3 and 4 you can also use the constants ARECeiver, BRECeiver, CRECeiver and DRECeiver, respectively.
Default unit: n/a

<Att> Attenuation factor for the received wave.
0 to 30 dB in steps of 10 dB.
*RST: 0 dB
Default unit: dB

Example: `POW:ATT AREC, 10`
Set an attenuation factor of 10 dB for the waves received at test port 1 and channel no. 1. The other test ports and channels are not affected.

Manual operation: See "Step Attenuator" on page 340

[SENSe<Ch>:]POWer:GAINcontrol <ReceiverName>, <Mode>
[SENSe<Ch>:]POWer:GAINcontrol? <ReceiverName>

Defines port-specific gain settings for the related channel.

Note:

- These settings will only take effect if `[SENSe<Ch>:]POWer:GAINcontrol:GLOBal` is set to `MANual` for channel <Ch>.
- Without the Extended power range option only the gain of the measurement receivers (the b-waves) can be statically set (to LNOise or LDIStortion); the reference receiver always uses automatic gain control (AUTO). With the option available, also the gain of the reference receivers (the a-waves) can be set statically.
- If you also want to distinguish between different sweep segments, use `[SENSe<Ch>:]SEGMent<Seg>:POWer:GAINcontrol` instead.

Suffix:
<Ch> Channel number

Parameters:
<Mode> AUTO | LDIStortion | LNOise

AUTO
automatic/adaptive gain control (AGC) according to the RF input level

LDIStortion
fixed, small IF gain, for high input levels

LNOise
fixed, large IF gain, for low input levels

*RST: LDIStortion

Parameters for setting and query:
<ReceiverName> String parameter defining the wave(s) followed by the drive port; see examples below and Table 7-3.

Example:
`*RST; :SENSe:POWer:GAINcontrol:GLOBal MANual`
Activate independent GC settings for each drive port.
`SENSe:POWer:GAINcontrol 'B2D1', LNO`
Set the GC for the received wave b_2 (port 2) to "Low Noise". The setting applies while port 1 is the drive port.
`SENSe:POWer:GAINcontrol 'A2B2B3D2', LNO`
Set the GC mode for the waves a_2, b_2, and b_3 to "Low Noise". The setting applies while port 2 is the drive port.
`SENSe:POWer:GAINcontrol? 'B1D2'`
Query the GC setting for the wave b_1 while port 2 is the drive port. The response is `1, LDIS` (default setting).

Manual operation: See "Drive-port specific settings" on page 702

[SENSe<Ch>:]POWer:GAINcontrol:ALL <Mode>

Applies the same manual gain control (GC) <Mode> to all a and b wave receivers.

Note:

- Without the Extended power range option only the gain of the measurement receivers (the b waves) can be statically set (to LNOise or LDIStortion); the reference receiver always uses automatic gain control (AUTO). With the option available, also the gain of the reference receivers (the a waves) can be set statically.
- If you also want to distinguish between different sweep segments, use `[SENSe<Ch>:]SEGMent<Seg>:POWer:GAINcontrol:ALL` instead.

Suffix:
<Ch> Channel number

Setting parameters:
<Mode> AUTO | LDIStortion | LNOise

AUTO
automatic/adaptive gain control (AGC) according to the RF input level

LDIStortion
fixed, small IF gain, for high input levels

LNOise
fixed, large IF gain, for low input levels

*RST: LDIStortion

Example: `*RST; :SENSe:POWer:GAINcontrol:GLOBal MANual`
Enable the manual configuration of independent GC settings for each drive port.
`SENSe:POWer:GAINcontrol:ALL LNO`
Set the GC mode of all a and b wave receivers to "Low Noise".

Usage: Setting only

Manual operation: See "Set All Items to ..." on page 703

[SENSe<Ch>:]POWer:GAINcontrol:GLOBal <GCModesGlobal>

Globally configures the gain control (GC) in all receive paths (measurement receivers, b-waves) for all analyzer ports or enables port-specific gain control configuration.

Suffix:
<Ch> Channel number

Parameters:
<GCModesGlobal> LNOise | LDIStortion | AUTO | MANual

AUTO
automatic/adaptive gain control (AGC) according to the RF input level

LDIStortion
fixed, small IF gain, for high input levels

LNOise
fixed, large IF gain, for low input levels

MANual
Enables the manual configuration of independent AGC settings for each segment, drive port and receiver. See `[SENSe<Ch>:]POWer:GAINcontrol` and `[SENSe<Ch>:]SEGMent<Seg>:POWer:GAINcontrol`.

*RST: LDIS

Example: See `[SENSe<Ch>:]POWer:GAINcontrol`

Manual operation: See "AGC Mode" on page 701

[SENSe<Ch>:]POWer:LOSuppress[:STATe] <Boolean>

Activates/deactivates "LO Suppression" for channel <Ch>.

"LO Suppression" lowers the LO power of all receivers and thus the power leaking out of the test ports.

R&S ZNB with recent synthesizer boards perform "LO Suppression" automatically; activation is not required and returns an error.

Suffix:
<Ch> Channel number

Parameters:
<Boolean>

OFF (0)
LO suppression active

ON (1)
LO suppression inactive

*RST: OFF

Manual operation: See "LO Suppression" on page 700

7.3.14.15 [SENSe:]ROSCillator...

The `[SENSe:]ROSCillator...` commands control the frequency reference signal.

[SENSe:]ROSCillator:EXTernal:FREQuency 1431
[SENSe:]ROSCillator[:SOURce] 1432

[SENSe:]ROSCillator:EXTernal:FREQuency <ExtRefFreq>

Specifies or queries the frequency of the external reference oscillator.

Parameters:
<ExtRefFreq> Frequency of the external reference clock signal.

Range: See the data sheet of your analyzer.
*RST: 10 MHz
Default unit: Hz

Example: `ROSC EXT`
Select the external reference clock as clock source.
`ROSC:EXT:FREQ 10MHz`
Specify the frequency of the external reference clock.
`ROSC:EXT:FREQ?`
Query the frequency of the external reference oscillator. The response is 10000000 Hz.

Manual operation: See "Ext Frequency" on page 842

[SENSe:]ROSCillator[:SOURce] <FreqRef>

Selects the source of the reference oscillator signal.

Parameters:

<FreqRef> INTernal | EXTernal

INTernal – internal 10 MHz reference oscillator
EXTernal – external reference clock
The frequency of the external reference clock can be defined using `[SENSe:]ROSCillator:EXTernal:FREQuency`.

*RST: INTernal

Example: See `[SENSe:]ROSCillator:EXTernal:FREQuency` on page 1431.

Manual operation: See "Internal/External" on page 842

7.3.14.16 [SENSe:]SEGMent...

The `[SENSe:]SEGMent<Seg>...` commands define all channel settings for a segmented frequency sweep. A segmented sweep is activated via `[SENSe<Ch>:]SWEep:TYPE SEGMent`.

The commands in this subsystem do not accept the step parameters UP and DOWN. Numeric values can be entered directly or using the DEFault, MINimum, MAXimum parameters.

[SENSe<Ch>:]SEGMent<Seg>:ADD....1433
[SENSe<Ch>:]SEGMent<Seg>:BWIDth[:RESolution]....1434
[SENSe<Ch>:]SEGMent<Seg>:BWIDth[:RESolution]:CONTrol....1434
[SENSe<Ch>:]SEGMent<Seg>:BWIDth[:RESolution]:SELect....1435
[SENSe<Ch>:]SEGMent<Seg>:BWIDth[:RESolution]:SELect:CONTrol....1435
[SENSe<Ch>:]SEGMent:COUNt?....1436
[SENSe<Ch>:]SEGMent<Seg>:DEFine....1437
[SENSe<Ch>:]SEGMent<Seg>:DEFine:SELect....1438
[SENSe<Ch>:]SEGMent<Seg>:DELete:ALL....1439
[SENSe<Ch>:]SEGMent<Seg>:DELete[:DUMMy]....1439
[SENSe<Ch>:]SEGMent<Seg>:FREQuency:CENTer?....1440
[SENSe<Ch>:]SEGMent<Seg>:FREQuency:SPAN?....1440

[SENSe<Ch>:]SEGMent<Seg>:FREQuency:STARt....1440
[SENSe<Ch>:]SEGMent<Seg>:FREQuency:STOP....1440
[SENSe<Ch>:]SEGMent<Seg>:LO....1441
[SENSe<Ch>:]SEGMent<Seg>:NAME....1441
[SENSe<Ch>:]SEGMent<Seg>:FREQuency:STEP....1441
[SENSe<Ch>:]SEGMent<Seg>:INSert....1442
[SENSe<Ch>:]SEGMent<Seg>:INSert:SELect....1444
[SENSe<Ch>:]SEGMent<Seg>:PORT<PortId>:POWer[:LEVel]....1444
[SENSe<Ch>:]SEGMent<Seg>:PORT<PortId>[:STATe]....1445
[SENSe<Ch>:]SEGMent<Seg>:POWer:GAINcontrol....1445
[SENSe<Ch>:]SEGMent<Seg>:POWer:GAINcontrol:ALL....1447
[SENSe<Ch>:]SEGMent<Seg>:POWer:GAINcontrol:CONTrol....1448
[SENSe<Ch>:]SEGMent<Seg>:POWer[:LEVel]....1448
[SENSe<Ch>:]SEGMent:POWer[:LEVel]:CONTrol....1449
[SENSe<Ch>:]SEGMent<Seg>[:STATe]....1450
[SENSe<Ch>:]SEGMent<Seg>:SWEep:DETector:TIME....1450
[SENSe<Ch>:]SEGMent<Seg>:SWEep:DETector:TIME:CONTrol....1450
[SENSe<Ch>:]SEGMent<Seg>:SWEep:DWELl....1450
[SENSe<Ch>:]SEGMent<Seg>:SWEep:DWELl:CONTrol....1451
[SENSe<Ch>:]SEGMent<Seg>:SWEep:GENeration....1452
[SENSe<Ch>:]SEGMent<Seg>:SWEep:POINts....1453
[SENSe<Ch>:]SEGMent<Seg>:SWEep:TIME....1453
[SENSe<Ch>:]SEGMent<Seg>:SWEep:TIME:CONTrol....1454
[SENSe<Ch>:]SEGMent<Seg>:SWEep:TIME:SUM?....1455

[SENSe<Ch>:]SEGMent<Seg>:ADD

Inserts a new sweep segment using default channel settings ("Insert New Segment"). The added segment covers the frequency interval between the maximum frequency of the existing sweep segments and the stop frequency of the entire sweep range.

Tip: Use `[SENSe<Ch>:]SEGMent<Seg>:INSert` to create a segment with specific channel settings.

Suffix:

<Ch> Channel number

<Seg> Sweep segment number. Segment numbers must be sequential. If n segments exist already, the added segment must have the segment number n+1.

Example:
```
SEGM:INS 1MHZ, 1.5MHZ, 111, -21DBM, 0.5S, 0, 10KHZ
```
Create a sweep segment with a sweep range between 1.0 MHz and 1.5 MHz.
```
SEGM2:ADD
```
Create a second sweep segment. The frequency range of the second segment will be between 1.5 MHz and the maximum frequency of the analyzer.

Usage: Event

Manual operation: See "Add / Insert / Delete / Delete All" on page 547

[SENSe<Ch>:]SEGMent<Seg>:BWIDth[:RESolution] <ResBandwidth>

Defines the IF bandwidth of the analyzer (measurement bandwidth) in sweep segment no. <Seg>. Values between 1 Hz and 1 MHz can be set. Option R&S ZNB-K17 enables receiver bandwidths up to 10 MHz (see Section 4.7.4, "Receiver bandwidth 10 MHz", on page 243).

Bandwidths can be set in 1 – 1.5 – 2 – 3 – 5 – 7 steps. The analyzer rounds up any entered value between these steps and rounds down values exceeding the maximum bandwidth.

At the same time, the command activates separate bandwidth setting in all sweep segments ([SENSe<Ch>:]SEGMent<Seg>:BWIDth[:RESolution]:CONTrol ON).

Suffix:

<Ch> Channel number

<Seg> Sweep segment number

Parameters:

<ResBandwidth> IF bandwidth

Range: See above
Increment: 1-1.5-2-3-5-7 steps
*RST: 10 kHz
Default unit: Hz

Example: See [SENSe<Ch>:]SEGMent<Seg>:BWIDth[:RESolution]:CONTrol

Manual operation: See "Optional Columns" on page 550

[SENSe<Ch>:]SEGMent<Seg>:BWIDth[:RESolution]:CONTrol <Boolean>

Selects common or independent "Meas. Bandwidth" settings for the sweep segments.

Suffix:

<Ch> Channel number

<Seg> Sweep segment number. This suffix is ignored; the setting controls the whole segmented sweep.

Parameters:

<Boolean> ON – use independent bandwidth settings, to be defined via [SENSe<Ch>:]SEGMent<Seg>:BWIDth[:RESolution].
OFF – reset the bandwidth in all sweep segments to the bandwidth for unsegmented sweeps, defined via [SENSe<Ch>:]BWIDth[:RESolution]. ON will not restore the previous values.
The parameter is automatically switched to ON when a bandwidth is entered using [SENSe<Ch>:]SEGMent<Seg>:BWIDth[:RESolution].

*RST: OFF

Example: `*RST; :SENS:SEGM:ADD`
Create a new sweep segment no. 1 in channel no. 1 using default settings and thus 10 kHz measurement bandwidth.
`SEGM:BWID 1 MHZ`
Increase the IF bandwidth to 1 MHz.
`SEGM:BWID:CONT OFF`
Couple the bandwidths in all segments and reset the bandwidth in segment no. 1 to the initial value.

Manual operation: See "Optional Columns" on page 550

[SENSe<Ch>:]SEGMent<Seg>:BWIDth[:RESolution]:SELect <Selectivity>

Defines the "Selectivity" of the IF filter used in sweep segment no. `<Seg>`. At the same time, the command activates individual selectivity settings for all sweep segments (`[SENSe<Ch>:]SEGMent<Seg>:BWIDth[:RESolution]:SELect:CONTrol ON`).

Suffix:
<Ch> Channel number

<Seg> Sweep segment number

Parameters:
<Selectivity> NORMal | MEDium | HIGH | NFIGure

NORMal - IF filter with normal selectivity and shortest settling time.
MEDium - IF filter with steeper edges and longer settling time.
HIGH - IF filter with highest selectivity but longest settling time.

Example: See `[SENSe<Ch>:]SEGMent<Seg>:BWIDth[:RESolution]:SELect:CONTrol`

Manual operation: See "Optional Columns" on page 550

[SENSe<Ch>:]SEGMent<Seg>:BWIDth[:RESolution]:SELect:CONTrol <Boolean>

Selects common or independent "Selectivity" settings for the individual sweep segments.

Suffix:
<Ch> Channel number

<Seg> Sweep segment number. This suffix is ignored; the setting controls the whole segmented sweep.

Parameters:

<Boolean> ON - use independent selectivity settings, to be defined via `[SENSe<Ch>:]SEGMent<Seg>:BWIDth[:RESolution]:SELect`.
OFF - reset the selectivity in all sweep segments to the selectivity for unsegmented sweeps, defined via `[SENSe<Ch>:]BWIDth[:RESolution]:SELect`. ON will not restore the previous values.
The parameter is automatically switched to ON when a selectivity is entered using `[SENSe<Ch>:]SEGMent<Seg>:BWIDth[:RESolution]:SELect`.

*RST: OFF

Example:

```
*RST; :SEGM:ADD
```
Create a new sweep segment no. 1 in channel no. 1 using default settings and thus NORMal selectivity.
```
SEGM:BWID:SEL HIGH
```
Change the selectivity to HIGH.
```
BWID:SEL?
```
Query the (default) selectivity for unsegmented sweeps. The response is NORM.
```
SEGM:BWID:SEL:CONT OFF
```
Couple the selectivities in all segments and reset the selectivity in segment no. 1 to the unsegmented value NORMal.

Manual operation: See "Optional Columns" on page 550

[SENSe<Ch>:]SEGMent:COUNt?

Returns the number of sweep segments in the channel including all segments that are switched off (`[SENSe<Ch>:]SEGMent<Seg>[:STATe] OFF`).

Suffix:

<Ch> Channel number

Example:

```
SEGM:ADD
```
Create a new sweep segment no. 1 in channel no. 1 using default settings.
```
SEGM OFF
```
Disable the measurement in the created sweep segment.
```
SEGM:COUN?
```
Query the number of segments.

Usage: Query only

Manual operation: See "Table Columns" on page 546

[SENSe<Ch>:]SEGMent<Seg>:DEFine <StartFreq>, <StopFreq>, <Points>, <Power>, <SegmentTime>|<MeasDelay>, <Unused>, <MeasBandwidth>[, <LO>[, <Selectivity>[, <FreqSweepMode>]]]]

Creates or re-defines a sweep segment no. <Seg> with specific channel settings.

Entry of the first seven numeric parameters is mandatory; no default values are provided. All settings except <LO> can be changed for existing segments using other commands of the [SENSe<Ch>:]SEGMent<Seg>... subsystem.

Note: Use [SENSe<Ch>:]SEGMent<Seg>:ADD to create a segment with default channel settings. Use [SENSe<Ch>:]SEGMent<Seg>:INSert (no query) to insert a new segment into the current segment list.

Suffix:

<Ch> Channel number

<Seg> Sweep segment number. Segment numbers must be sequential. The specified segment number must be smaller or equal to the number of existing segments plus 1.
If segment number <Seg> already exists, it is replaced by the new segment.

Parameters:

<StartFreq> Start frequency of the segment; see [SENSe<Ch>:]SEGMent<Seg>:FREQuency:STARt.

Default unit: Hz

<StopFreq> Stop frequency of the segment; see [SENSe<Ch>:]SEGMent<Seg>:FREQuency:STOP.

Default unit: Hz

<Points> Number of sweep points in the segment. See [SENSe<Ch>:]SEGMent<Seg>:SWEep:POINts.

<Power> Internal source power in the segment. See [SENSe<Ch>:]SEGMent<Seg>:POWer[:LEVel].

Default unit: dBm

<SegmentTime> Duration of the sweep in the segment. See [SENSe<Ch>:]SEGMent<Seg>:SWEep:TIME. In the setting [SENSe<Ch>:]SEGMent<Seg>:INSert:SELect DWELl, this parameter is replaced by <MeasDelay>.

Range: Depending on other channel settings. AUTO activates automatic sweep time setting in the segment, which is equivalent to the minimum sweep time possible.

Default unit: s

<MeasDelay> Delay for each partial measurement in the segment. See [SENSe<Ch>:]SEGMent<Seg>:SWEep:DWELl. In the setting [SENSe<Ch>:]SEGMent<Seg>:INSert:SELect SWTime, this parameter is replaced by <SegmentTime>.

<Unused> Ignored parameter (for compatibility with R&S ZVR analyzers). Should be set to the default value 0.

<MeasBandwidth> IF bandwidth in the segment. See `[SENSe<Ch>:]SEGMent<Seg>:BWIDth[:RESolution]`.

Default unit: Hz

<LO> POSitive | NEGative | AUTO

Position of the local oscillator frequency LO relative to the RF frequency. In remote control this parameter must be set when a sweep segment is created.
See `[SENSe<Ch>:]FREQuency:SBANd`.

<Selectivity> NORMal | MEDium | HIGH

Selectivity of the IF filter. See `[SENSe<Ch>:]SEGMent<Seg>:BWIDth[:RESolution]:SELect`.

<Mode> AUTO

Ignored.

<FreqSweepMode> STEPped | ANALog

"Stepped" mode (=`STEPped`, default) or swept mode (=`ANALog`). See Section 4.1.4.5, "Stepped vs. swept mode", on page 88.

Example:
```
SEGM:ADD
```
Create a new sweep segment no. 1 in channel no. 1 using default settings.
```
SEGM:DEF?
```
Query the channel settings for the new segment. A possible response is
```
1000000000,2000000000,51,-300,0.0044625,0,10000,AUTO,NORM,AUTO,STEP.
```

Manual operation: See "Add / Insert / Delete / Delete All" on page 547

[SENSe<Ch>:]SEGMent<Seg>:DEFine:SELect <TimeRef>

Defines whether the sweep time of a new segment, i.e. numeric parameter no. 5 of the command `[SENSe<Ch>:]SEGMent<Seg>:DEFine`, is entered as a segment sweep time ("Segment Time") or as a measurement delay ("Meas Delay").

Suffix:

<Ch> Channel number

<Seg> Sweep segment number

Parameters:

<TimeRef> SWTime | DWELl

SWTime - use segment sweep time.
DWELl - use measurement delay.

Example: SEGM1:DEF:SEL DWEL
Select the measurement delay to determine the sweep time in a new sweep segment no. 1.
SEGM1:DEF 1MHZ, 1.5MHZ, 111, -21DBM, 0.01S, 0, 10KHZ
Create a sweep segment with a sweep range between 1.0 MHz and 1.5 MHz and a measurement delay of 10 ms.
SEGM1:SWE:TIME?
Query the sweep time in the new segment.

Manual operation: See "Optional Columns" on page 550

[SENSe<Ch>:]SEGMent<Seg>:DELete:ALL

Deletes all sweep segments in the channel. [SENSe<Ch>:]SEGMent<Seg>:DELete[:DUMMy] deletes a single segment.

Suffix:
<Ch> Channel number

<Seg> Sweep segment number. This suffix is ignored; the command deletes all segments.

Example: SEGM:ADD
Create a new sweep segment no. 1 in channel no. 1 using default settings.
SEGM:DEL:ALL
Delete the created segment and all segments in the channel created before.

Usage: Event

Manual operation: See "Add / Insert / Delete / Delete All" on page 547

[SENSe<Ch>:]SEGMent<Seg>:DELete[:DUMMy]

Deletes the specified (single) sweep segment. [SENSe<Ch>:]SEGMent<Seg>:DELete:ALL deletes all segments in the channel.

Suffix:
<Ch> Channel number

<Seg> Sweep segment number

Example: SEGM:ADD
Create a new sweep segment no. 1 in channel no. 1 using default settings.
SEGM:DEL
Delete the created segment.

Manual operation: See "Add / Insert / Delete / Delete All" on page 547

[SENSe<Ch>:]SEGMent<Seg>:FREQuency:CENTer?
[SENSe<Ch>:]SEGMent<Seg>:FREQuency:SPAN?

These commands return the center frequency and the span (width) of sweep segment no. <Seg>.

Suffix:
<Ch> Channel number

<Seg> Sweep segment number

Example: SEGM:INS 1MHZ, 1.5MHZ, 111, -21DBM, 0.5S, 0, 10KHZ
Create a sweep segment with a sweep range between 1.0 MHz and 1.5 MHz.
SEGM:FREQ:CENT? SPAN?
Query the center frequency and span of the created segment. The response is 1250000;500000.

Usage: Query only

Manual operation: See "Table Columns" on page 546

Note: The frequency range of the sweep segment can be changed via [SENSe<Ch>:]SEGMent<Seg>:FREQuency:STARt and [SENSe<Ch>:]SEGMent<Seg>:FREQuency:STOP.

[SENSe<Ch>:]SEGMent<Seg>:FREQuency:STARt <StartFreq>
[SENSe<Ch>:]SEGMent<Seg>:FREQuency:STOP <StopFreq>

These commands define the start and stop frequency of sweep segment no. <Seg>.

The sweep segments must be within the frequency range of the R&S ZNB; see Table 7-18.

Suffix:
<Ch> Channel number

<Seg> Sweep segment number

Parameters:
<StopFreq> Start or stop frequency of the sweep.

Increment: 0.1 kHz
Default unit: Hz

Example: SEGM:INS 1MHZ, 1.5MHZ, 111, -21DBM, 0.5S, 0, 10KHZ
Create a sweep segment with a sweep range between 1.0 MHz and 1.5 MHz.
SEGM:FREQ:STAR?; STOP?
Query the start and stop frequency of the created segment. The response is 1000000;1500000.

Manual operation: See "Table Columns" on page 546

Note: If the start frequency entered is greater than the current stop frequency, the stop frequency is set to the start frequency. If the stop frequency entered is smaller than the current start frequency, the start frequency is set to the stop frequency.

[SENSe<Ch>:]SEGMent<Seg>:LO <Sideband>

For a segmented sweep in channel <Ch>, this command defines the position of the local oscillator frequency LO relative to the RF frequency in sweep segment no. `<Seg>`.

Suffix:
<Ch> Channel number.

<Seg> Sweep segment number

Parameters:
<Sideband> POSitive | NEGative | AUTO

See `[SENSe<Ch>:]FREQuency:SBANd`.

Manual operation: See "Optional Columns" on page 550

[SENSe<Ch>:]SEGMent<Seg>:NAME <SegmentName>
[SENSe<Ch>:]SEGMent<Seg>:NAME? <SegmentName>

For a segmented sweep in channel <Ch>, this command defines the name of segment no. `<Seg>`.

Suffix:
<Ch> Channel number.

<Seg> Sweep segment number

Parameters for setting and query:
<SegmentName> Name of segment no. <Seg>

Manual operation: See "Optional Columns" on page 550

[SENSe<Ch>:]SEGMent<Seg>:FREQuency:STEP <Size>

For segmented sweeps in channel `<Ch>`, this command defines the frequency step size of segment `<Seg>`.

Suffix:
<Ch> Channel number

<Seg> Segment number

Parameters:
<Size> Frequency step size

Default unit: Hz

Manual operation: See "Table Columns" on page 546

[SENSe<Ch>:]SEGMent<Seg>:INSert <StartFreq>, <StopFreq>, <Points>, <Power>, <SegmentTime>|<MeasDelay>, <Unused>, <MeasBandwidth>[, <LO>[, <Selectivity>[, <Mode>[, <FreqSweepMode>]]]]

Adds a new sweep segment no. <Seg> with specific channel settings.

Entry of the first seven numeric parameters is mandatory; no default values are provided. All settings except <LO> can be changed for existing segments using other commands of the [SENSe<Ch>:]SEGMent<Seg>... subsystem.

Note: Use [SENSe<Ch>:]SEGMent<Seg>:ADD to create a segment with default channel settings. Use [SENSe<Ch>:]SEGMent<Seg>:DEFine to change or query all settings of an existing segment.

Suffix:

<Ch> Channel number

<Seg> Sweep segment number. Segment numbers must be sequential. The specified segment number must be smaller or equal to the number of existing segments plus 1.
If one or more sweep segments with segment numbers <Seg> or larger exist in the current channel, then all these existing segment numbers are incremented by 1 and the new segment is inserted as segment no. <Seg>.
Note that <Seg> defaults to 1, so [SENSe<Ch>:]SEGMent:INSert inserts a new segment 1, shifting up the existing segments by one position.

Parameters:

<StartFreq> Start frequency of the segment. See [SENSe<Ch>:]SEGMent<Seg>:FREQuency:STARt.
Default unit: Hz

<StopFreq> Stop frequency of the segment; see [SENSe<Ch>:]SEGMent<Seg>:FREQuency:STOP.
Default unit: Hz

<Points> Number of sweep points in the segment. See [SENSe<Ch>:]SEGMent<Seg>:SWEep:POINts.

<Power> Internal source power in the segment. See [SENSe<Ch>:]SEGMent<Seg>:POWer[:LEVel].
Default unit: dBm

<SegmentTime> Duration of the sweep in the segment. See [SENSe<Ch>:]SEGMent<Seg>:SWEep:TIME. In the setting [SENSe<Ch>:]SEGMent<Seg>:INSert:SELect DWELl, this parameter is replaced by <MeasDelay>.

Range: Depending on other channel settings. AUTO activates automatic sweep time setting in the segment, which is equivalent to the minimum sweep time possible.

Default unit: s

<MeasDelay> Delay for each partial measurement in the segment. See [SENSe<Ch>:]SEGMent<Seg>:SWEep:DWELl. In the setting [SENSe<Ch>:]SEGMent<Seg>:INSert:SELect SWTime, this parameter is replaced by <SegmentTime>.

<Unused> Ignored parameter (for compatibility with R&S ZVR analyzers). Should be set to the default value 0.

<MeasBandwidth> IF bandwidth in the segment. See [SENSe<Ch>:]SEGMent<Seg>:BWIDth[:RESolution].

Default unit: Hz

<LO> POSitive | NEGative | AUTO

Position of the local oscillator frequency LO relative to the RF frequency. In remote control this parameter must be set when a sweep segment is created.
See [SENSe<Ch>:]FREQuency:SBANd.

<Selectivity> NORMal | MEDium | HIGH

Selectivity of the IF filter. See [SENSe<Ch>:]SEGMent<Seg>:BWIDth[:RESolution]:SELect.

<Mode> AUTO

Ignored.

<FreqSweepMode> STEPped | ANALog

"Stepped" mode (STEPped, default) or "Swept" mode (=ANALog). See Section 4.1.4.5, "Stepped vs. swept mode", on page 88.

Example:

```
SEGM:INS 1MHZ, 1.5MHZ, 111, -21DBM, 0.5S, 0, 10KHZ
```

Create a sweep segment with a sweep range between 1.0 MHz and 1.5 MHz.

```
SEGM2:ADD
```

Create a second sweep segment. The frequency range of the second segment will be between 1.5 MHz and the maximum frequency of the analyzer.

Manual operation: See "Add / Insert / Delete / Delete All" on page 547

[SENSe<Ch>:]SEGMent<Seg>:INSert:SELect <TimeRef>

Defines whether the sweep time of a new segment, i.e. numeric parameter no. 5 of the command `[SENSe<Ch>:]SEGMent<Seg>:INSert`, is entered as a segment sweep time ("Segment Time") or as a measurement delay ("Meas Delay").

Suffix:

<Ch> Channel number

<Seg> Sweep segment number

Parameters:

<TimeRef> SWTime | DWELl

SWTime - use segment sweep time.
DWELl - use measurement delay.

*RST: SWTime

Example: `SEGM1:INS:SEL DWEL`
Select the meas. delay to determine the sweep time in a new sweep segment no. 1.
`SEGM1:INS 1MHZ, 1.5MHZ, 111, -21DBM, 0.01S, 0, 10KHZ`
Create a sweep segment with a sweep range between 1.0 MHz and 1.5 MHz and a meas. delay of 10 ms.
`SEGM1:SWE:TIME?`
Query the sweep time in the new segment.

Manual operation: See "Optional Columns" on page 550

[SENSe<Ch>:]SEGMent<Seg>:PORT<PortId>:POWer[:LEVel] <IntSourcePortPow>

For a segmented sweep in channel <Ch>, this command defines a port-specific power level for port <PortId> in sweep segment <Seg>.

Note

- Can only be set if `[SENSe<Ch>:]SEGMent<Seg>:PORT<PortId>[:STATe]` is ON (for this channel, segment and port).
- Only takes effect if `[SENSe<Ch>:]SEGMent:POWer[:LEVel]:CONTrol` is set to ON

Suffix:

<Ch> Channel number

<Seg> Sweep segment number

<PortId> Port number

Parameters:

<IntSourcePortPow> Power level for port <PortId> in sweep segment <Seg> of channel <Ch>

Default unit: dBm

Manual operation: See "Optional Columns" on page 550

[SENSe<Ch>:]SEGMent<Seg>:PORT<PortId>[:STATe] <Boolean>

For a segmented sweep in channel <Ch>, this command enables setting a port-specific power level for port <PortId> in segment <Seg>.

The port-specific power level can then be set using `[SENSe<Ch>:]SEGMent<Seg>:POWer[:LEVel]`.

Note

- Port-specific power levels only apply if segment-specific power levels are enabled (`[SENSe<Ch>:]SEGMent:POWer[:LEVel]:CONTrol ON`).
- If segment-specific source power levels are disabled (`[SENSe<Ch>:]SEGMent:POWer[:LEVel]:CONTrol OFF`), all segment- and port-specific levels are reset to the channel base power.

Suffix:

<Ch> Channel number

<Seg> Sweep segment number

<PortId> Port number

Parameters:

<Boolean>

OFF (0)
No port-specific power level for port <PortId> in segment <Seg> of channel <Ch>.
If `[SENSe<Ch>:]SEGMent:POWer[:LEVel]:CONTrol` is set to ON, then the segment power level `[SENSe<Ch>:]SEGMent<Seg>:POWer[:LEVel]` is used.
Otherwise the channel base power `SOURce<Ch>:POWer<PhyPt>[:LEVel][:IMMediate][:AMPLitude]` is used.

ON (1)
If `[SENSe<Ch>:]SEGMent:POWer[:LEVel]:CONTrol` is set to ON, then the port-specific power level `[SENSe<Ch>:]SEGMent<Seg>:PORT<PortId>:POWer[:LEVel]` is used for port <PortId> in segment <Seg> of channel <Ch>.
Otherwise the channel base power `SOURce<Ch>:POWer<PhyPt>[:LEVel][:IMMediate][:AMPLitude]` is used.

*RST: OFF

Manual operation: See "Optional Columns" on page 550

[SENSe<Ch>:]SEGMent<Seg>:POWer:GAINcontrol <ReceiverName>, <Mode>
[SENSe<Ch>:]SEGMent<Seg>:POWer:GAINcontrol? <ReceiverName>

Defines port- and segment-specific IF gain settings.

These settings apply if and only if segmented gain control (GC) is enabled (see `[SENSe<Ch>:]SEGMent<Seg>:POWer:GAINcontrol:CONTrol`).

Note:

- Without the Extended power range option only the gain of the measurement receivers (the b-waves) can be statically set (to LNOise or LDIStortion); the reference receiver always uses automatic gain control (AUTO). With the option available, also the gain of the reference receivers (the a-waves) can be set statically.
- Disable segmented GC and use `[SENSe<Ch>:]POWer:GAINcontrol` instead, if no distinction between different sweep segments shall be made.

Suffix:

<Ch> Channel number

<Seg> Sweep segment number

Setting parameters:

<Mode> AUTO | LDIStortion | LNOise

AUTO
automatic/adaptive gain control (AGC) according to the RF input level

LDIStortion
fixed, small IF gain, for high input levels

LNOise
fixed, large IF gain, for low input levels

*RST: LDIStortion

Parameters for setting and query:

<ReceiverName> String parameter defining the wave(s) followed by the drive port; see examples below and Table 7-3.

Example:

```
*RST; :SENSe:POWer:GAINcontrol:GLOBal MANual
```
Activate independent GC settings for each drive port.
```
SENSe:SEGMent1:ADD; :SENSe:SWEep:TYPE SEGMent
```
Create a new sweep segment no. 1 and activate segmented sweep type.
```
SENSe:SEGMent1:POWer:GAINcontrol:CONTrol ON
```
Enable segmented GC.
```
SENSe:SEGMent1:POWer:GAINcontrol 'B2D1', LNO
```
Set the ACG for the received wave b_2 (port 2) in sweep segment no. 1 to "Low Noise". The setting applies while port 1 is the drive port.
```
SENSe:SEGMent1:POWer:GAINcontrol 'A2B2B3D2',
LNO
```
Set the ACG for the waves a_2, b_2, and b_3 in sweep segment no. 1 to "Low Noise". The setting applies while port 2 is the drive port.
```
SENSe:SEGMent1:POWer:GAINcontrol? 'B1D2'
```
Query the ACG setting for the wave b_1 in sweep segment no. 1 while port 2 is the drive port. The response is `1, LDIS` (default setting).

Manual operation: See "Range" on page 702

[SENSe<Ch>:]SEGMent<Seg>:POWer:GAINcontrol:ALL <Mode>

For the selected segment, this command applies the same manual gain control (GC) <Mode> to all a and b wave receivers.

These settings apply if and only if segmented GC is enabled (see `[SENSe<Ch>:]SEGMent<Seg>:POWer:GAINcontrol:CONTrol`).

Note:

- Without the Extended power range option only the gain of the measurement receivers (the b waves) can be statically set (to LNOise or LDIStortion); the reference receiver always uses automatic gain control (AUTO). With the option available, also the gain of the reference receivers (the a waves) can be set statically.
- Disable segmented GC and use `[SENSe<Ch>:]POWer:GAINcontrol` or `[SENSe<Ch>:]POWer:GAINcontrol:ALL` instead, if no distinction between different sweep segments shall be made.

Suffix:

<Ch> Channel number

<Seg> Sweep segment number

Setting parameters:

<Mode> AUTO | LDIStortion | LNOise

AUTO
automatic/adaptive gain control (AGC) according to the RF input level

LDIStortion
fixed, small IF gain, for high input levels

LNOise
fixed, large IF gain, for low input levels

*RST: LDIStortion

Example:

```
*RST; :SENSe:POWer:GAINcontrol:GLOBal MANual
```
Enable the manual configuration of independent GC settings for each drive port.

```
:SENSe:SEGMent1:ADD; :SENSe:SWEep:TYPE SEGMent
```
Create a new sweep segment no. 1 and activate segmented sweep type.

```
SENSe:SEGMent1:POWer:GAINcontrol:CONTrol ON
```
Enable segmented GC.

```
SENSe:SEGMent1:POWer:GAINcontrol:ALL LNO
```
For sweep segment no. 1, set the GC mode of all a and b wave receivers to "Low Noise". Segmented GC is enabled automatically.

Usage: Setting only

Firmware/software: V2.75 or higher

Manual operation: See "Range" on page 702

[SENSe<Ch>:]SEGMent<Seg>:POWer:GAINcontrol:CONTrol <Boolean>

Defines whether common or independent gain control (GC) settings shall be used for the individual sweep segments.

Suffix:
<Ch> Channel number

<Seg> Sweep segment number. This suffix is ignored.

Parameters:
<Boolean> ON – use independent GC settings, to be defined via `[SENSe<Ch>:]SEGMent<Seg>:POWer:GAINcontrol`.
OFF – reset the GC mode in all sweep segments to the GC mode for unsegmented sweeps, defined via `[SENSe<Ch>:]POWer:GAINcontrol` or `[SENSe<Ch>:]POWer:GAINcontrol:GLOBal`. ON will not restore the previous values.

*RST: ON

Example: See `[SENSe<Ch>:]SEGMent<Seg>:POWer:GAINcontrol`

Manual operation: See "Segmented AGC" on page 701

[SENSe<Ch>:]SEGMent<Seg>:POWer[:LEVel] <IntSourcePow>

For a segmented sweep in channel <Ch>, this command sets or queries the default source power level for sweep segment no. `<Seg>`.

The default level can be overridden by port-specific source power levels, which can be enabled and defined using `[SENSe<Ch>:]SEGMent<Seg>:PORT<PortId>[:STATe] ON` and `[SENSe<Ch>:]SEGMent<Seg>:PORT<PortId>:POWer[:LEVel]`, respectively.

Note

- The set command automatically enables segment-specific source power levels (`[SENSe<Ch>:]SEGMent:POWer[:LEVel]:CONTrol ON`).
- If segment-specific source power levels are disabled (`[SENSe<Ch>:]SEGMent:POWer[:LEVel]:CONTrol OFF`), all segment- and port-specific levels are reset to the channel base power.

Suffix:
<Ch> Channel number

<Seg> Sweep segment number

Parameters:
<IntSourcePow> Internal source power

Range: -300 dBm to +200 dBm. The usable power range is frequency-dependent; refer to the data sheet.
*RST: -10 dBm
Default unit: dBm

Example: `SEGM:ADD`
Create a new sweep segment no. 1 in channel no. 1 using default settings and thus -10 dBm internal source power.
`SEGM:POW -20`
Decrease the power to -20 dBm.

Manual operation: See "Optional Columns" on page 550

[SENSe<Ch>:]SEGMent:POWer[:LEVel]:CONTrol <Boolean>

For a segmented sweep in channel <Ch>, this command enables or disables segment-specific source power settings.

Note that segment-specific source power settings are enabled automatically when a segment-specific source power level is set using `[SENSe<Ch>:]SEGMent<Seg>:POWer[:LEVel]`.

Suffix:
<Ch> Channel number

Parameters:
<Boolean>

ON
Segment-specific power settings enabled
Use `[SENSe<Ch>:]SEGMent<Seg>:POWer[:LEVel]` to set a (default) power level for all ports.
Or use `[SENSe<Ch>:]SEGMent<Seg>:PORT<PortId>[:STATe] ON` to enable a segment- and port-specific power level for port <PortId> and set its value using `[SENSe<Ch>:]SEGMent<Seg>:PORT<PortId>:POWer[:LEVel]`.

OFF
Segment-specific power settings disabled
For all segments and ports, the power level is set to the channel base power (`SOURce<Ch>:POWer<PhyPt>[:LEVel][:IMMediate][:AMPLitude]`).
Note that setting
`[SENSe<Ch>:]SEGMent<Seg>:POWer[:LEVel]:CONTrol`
to OFF also resets all previously defined segment- and port-specific power levels (see explanation for ON above). Setting it to ON does not restore them.

*RST: OFF

Example: `SEGM:ADD`
Create a new sweep segment no. 1 in channel no. 1 using default settings and thus -10 dBm internal source power.
`SEGM:POW -20`
Decrease the power to -20 dBm.
`SEGM:POW:CONT OFF`
Couple the powers in all segments and reset the power in segment no. 1 to the initial value.

Manual operation: See "Optional Columns" on page 550

[SENSe<Ch>:]SEGMent<Seg>[:STATe] <Boolean>

Activates or deactivates the sweep segment <Seg>. Sweep points belonging to inactive segments only are not measured.

Suffix:
<Ch> Channel number

<Seg> Sweep segment number

Parameters:
<Boolean> ON | OFF - activates or deactivates the measurement in sweep segment <Seg>.

*RST: ON

Example: SEGM:ADD
Create a new sweep segment no. 1 in channel no. 1 using default settings.
SEGM OFF
Disable the measurement in the created sweep segment.

Manual operation: See "Table Columns" on page 546

[SENSe<Ch>:]SEGMent<Seg>:SWEep:DETector:TIME <SegSweepDetectorTime>

Suffix:
<Ch>

<Seg>

Parameters:
<SegSweepDetectorTime> Default unit: s

[SENSe<Ch>:]SEGMent<Seg>:SWEep:DETector:TIME:CONTrol <Boolean>

Suffix:
<Ch>

<Seg>

Parameters:
<Boolean>

[SENSe<Ch>:]SEGMent<Seg>:SWEep:DWELl <MeasDelay>

Defines the delay time for each partial measurement in sweep segment no. <Seg> ("Meas. Delay"). If coupling of the segments is switched on ([SENSe<Ch>:]SEGMent<Seg>:SWEep:DWELl:CONTrol ON), the delay is valid for all sweep segments in the current channel.

Suffix:
<Ch> Channel number

<Seg> Sweep segment number

Parameters:

<MeasDelay> Measurement delay before each partial measurement. Changing the delay leaves the number of points unchanged but has an impact on the duration of the sweep (`[SENSe<Ch>:]SEGMent<Seg>:SWEep:TIME`).

Range: 0 s to 2500 s
*RST: 0 s
Default unit: s

Example:

```
SEGM:ADD
```
Create a new sweep segment no. 1 in channel no. 1 using default settings.
```
SEGM:SWE:DWEL 1 MS
```
Set the meas. delay in segment no. 1 to 1 ms.
```
SEGM:DEF? Response:
300000,8000000000,51,-300,0.056559,0,10000,POS,
NORM
```
Query the channel parameters for sweep segment 1. The response value for the segment sweep time (olive) implicitly contains the defined meas. delay.

Manual operation: See "Optional Columns" on page 550

[SENSe<Ch>:]SEGMent<Seg>:SWEep:DWELl:CONTrol <Boolean>

Selects common or independent "Meas. Delay" settings for the sweep segments.

Suffix:

<Ch> Channel number.

<Seg> Sweep segment number. This suffix is ignored; the command controls the whole segmented sweep.

Parameters:

<Boolean> ON – use independent delay settings, to be defined via `[SENSe<Ch>:]SEGMent<Seg>:SWEep:DWELl`.
OFF – reset the delay in all sweep segments to the delay for unsegmented sweeps, defined via `[SENSe<Ch>:]SWEep:DWELl`.
The parameter is automatically switched to ON when a meas. delay time is entered using `[SENSe<Ch>:]SEGMent<Seg>:SWEep:DWELl`.

*RST: OFF

Example:
```
SEGM:ADD
```
Create a new sweep segment no. 1 in channel no. 1 using default settings and thus 0 s meas. delay.
```
SEGM:SWE:DWELl 0.1
```
Increase the meas. delay to 0.1 s.
```
SEGM:SWE:DWELl:CONT OFF
```
Couple the meas. delay in all segments and reset the delay in segment no. 1 to the initial value of 0 s.

Manual operation: See "Optional Columns" on page 550

[SENSe<Ch>:]SEGMent<Seg>:SWEep:GENeration <Mode>

Sets/gets the segment-specific sweep modes (steppd or swept) for segmented frequency sweeps.

Overwrites `[SENSe<Ch>:]SWEep:GENeration`.

Note that swept mode is **not** supported on the following instruments:

- R&S ZNB43
- R&S ZNBT equipped with synth boards SYNTH13DDS_DS or SYNTH13DDS_L, either ex factory or via upgrade.

  These synth boards are shipped with:
  - R&S ZNBT8|26|40 with serial number ≥ 200000
  - R&S ZNBT20 variant 64 (order no. 1332.9002K64)

  Use `SYSTem:DFPRint?` to query hardware-related information such as the synth boards equipping of your instrument.

Suffix:

<Ch> Channel number

<Seg> Number of an existing sweep segment

Parameters:

<Mode> STEPped | ANALog

STEPped
Stepped mode

ANALog
Swept mode

Example:
```
*RST
SEGM:INS 1MHZ, 1.5MHZ, 111, -21DBM, 0.5S, 0, 10KHZ
```
Creates segment 1 with the given parameters and the default frequency sweep mode `STEP`.
```
SEGM:INS 3MHZ, 4.5MHZ, 111, -21DBM, 0.5S, 0, 10KHZ, POS, NORM, AUTO, ANAL
```
Inserts a new segment 1 with the given parameters, moving the previous segment 1 to segment 2. Hence
```
SENSe:SEGMent1:SWEep:GENeration?
```
returns `ANAL` and
```
SENSe:SEGMent2:SWEep:GENeration?
```
returns `STEP`.

Manual operation: See "Optional Columns" on page 550

[SENSe<Ch>:]SEGMent<Seg>:SWEep:POINts <SegPoint>

Defines the total number of measurement points in sweep segment no. `<Seg>`.

Suffix:
<Ch> Channel number

<Seg> Sweep segment number

Parameters:
<SegPoint> Number of points in the segment, up to the maximum number of sweep points N_{max} = *100001*. A value of 1 is allowed if start and stop frequencies are equal.

*RST: 51

Example:
```
SEGM:ADD
```
Create a new sweep segment no. 1 in channel no. 1 using default settings.
```
SEGM:SWE:POIN 401
```
Increase the number of points to 401.

Manual operation: See "Table Columns" on page 546

[SENSe<Ch>:]SEGMent<Seg>:SWEep:TIME <SegSweepTime>

Sets the duration of the sweep in sweep segment no. `<Seg>` ("Segment Time"). At the same time, the command activates separate sweep time setting in all sweep segments (`[SENSe<Ch>:]SEGMent<Seg>:SWEep:TIME:CONTrol ON`).

Suffix:
<Ch> Channel number

<Seg> Sweep segment number

Parameters:

<SegSweepTime> Segment time. The minimum segment time depends on the other channel settings, in particular on the number of points ([SENSe<Ch>:]SEGMent<Seg>:SWEep:POINts), the IF bandwidth ([SENSe<Ch>:]SEGMent<Seg>:BWIDth[:RESolution]) and the delay for each partial measurement ([SENSe<Ch>:]SEGMent<Seg>:SWEep:DWELl). The maximum is 1000 s. Changing the duration leaves the number of points unchanged but directly affects the delay.

Range: Minimum value to 1000 s
*RST: Minimum value, depending on the channel settings. This default value corresponds to automatic sweep time setting in manual control.
Default unit: s

Example:

```
SEGM:ADD
```
Create a new sweep segment no. 1 in channel no. 1.
```
SEGM:SWE:TIME 0.1
```
Increase the segment sweep time to 0.1 s.
```
SEGM:SWE:TIME:SUM?
```
Query the total duration of the segmented sweep. The response is 0.1, because there is only one sweep segment.

Manual operation: See "Optional Columns" on page 550

[SENSe<Ch>:]SEGMent<Seg>:SWEep:TIME:CONTrol <Boolean>

Selects common or independent "Segment Sweep Time" settings for the sweep segments.

Suffix:

<Ch> Channel number

<Seg> Sweep segment number. This suffix is ignored; the command controls the whole segmented sweep.

Parameters:

<Boolean> ON – use independent segment sweep time settings, to be defined via [SENSe<Ch>:]SEGMent<Seg>:SWEep:TIME.
OFF – reset the segment sweep time in all sweep segments to the segment sweep time for unsegmented sweeps, defined via [SENSe<Ch>:]SWEep:TIME. ON will not restore the previous values.
The parameter is automatically switched to ON when a segment sweep time is entered using [SENSe<Ch>:]SEGMent<Seg>:SWEep:TIME or if the channel settings in a sweep segment require a sweep time larger than the unsegmented sweep time.

*RST: OFF

Example: `SEGM:ADD`
Create a new sweep segment no. 1 in channel no. 1.
`SEGM:SWE:TIME 0.1`
Increase the segment sweep time to 0.1 s.
`SEGM:SWE:TIME:CONT OFF`
Couple the sweep times in all segments and reset the sweep time in segment no. 1 to the initial value.

Manual operation: See "Optional Columns" on page 550

[SENSe<Ch>:]SEGMent<Seg>:SWEep:TIME:SUM?

Returns the total duration of the segmented sweep, calculated as the sum of the sweep times of the individual segments (`[SENSe<Ch>:]SEGMent<Seg>:SWEep:TIME`).

Suffix:
<Ch> Channel number.

<Seg> Sweep segment number. This suffix is ignored; the command returns the sum of all segments.

Example: See `[SENSe<Ch>:]SEGMent<Seg>:SWEep:TIME`

Usage: Query only

Manual operation: See "Optional Columns" on page 550

7.3.14.17 [SENSe<Ch>:]SNPMeasure...

These commands support the functionality of the Section 4.7.8, "SNP assistant", on page 251 (R&S ZNB-K100).

[SENSe:]SNPMeasure:DISPlay:CMAP<pos>:RGB <Red>, <Green>, <Blue>

Defines the color scheme of the "SNP Assistant Overview".

Suffix:
<pos> Numeric identifier for the S-parameter state:
1 – Measured
2 – Unmeasured
3 – Imported
4 – Idealized

Parameters:
<Red>, <Green>, <Blue> RGB values
Range: 0 to 255

Options: R&S ZNB-K100

Manual operation: See "Define Colors" on page 425

[SENSe:]SNPMeasure:DISPlay:RST

Resets the color scheme of the "SNP Assistant Overview" to its factory default.

Usage: Event

Options: R&S ZNB-K100

Manual operation: See "Define Colors" on page 425

[SENSe:]SNPMeasure:DUT?

Displays the topology of the modeled DUT.

Lists the types, port numbers and port labels of the DUT structures:

- As returned by `[SENSe:]SNPMeasure:DUT:PART<pos>?`
- Separated by semicolons

Example: For the unmodified template "SFP +", `:SENSe1:SNPMeasure:DUT?` returns

```
D4,1,'1A_pos',2,'1B_pos',3,'1A_neg',4,'1B_neg';
D4,5,'1A_pos',6,'1B_pos',7,'1A_neg',8,'1B_neg
```

(line break at semicolon added for readability reasons)

Usage: Query only

Options: R&S ZNB-K100

Manual operation: See "DUT topology table" on page 413

[SENSe:]SNPMeasure:DUT:IMPort <Measurement SParam>, <Touchstone SParam>, <Touchstone File>

Imports S-parameter data to the current SNPA model.

Note that the frequency points of the loaded Touchstone file must be compatible with the sweep settings of the SNPA. Otherwise an error is raised.

Setting parameters:

<Measurement SParam> `'S<i><j>'` where `<i>` and `<j>` are DUT port numbers

<Touchstone SParam> `'S<k><l>'` where `<k>` and `<l>` select the S-parameter in the Touchstone file (by position, not by port name!)

<Touchstone File> String parameter specifying the name and directory of the Touchstone file.

Usage: Setting only

Options: R&S ZNB-K100

Manual operation: See "Data" on page 420

[SENSe:]SNPMeasure:DUT:INIT:IDEalize <SParamType>
[SENSe:]SNPMeasure:DUT:INIT:MEASure <SParamType>

Tells the SNPA to idealize or measure certain logical groups of S-parameters, i.e. to set their data states to "Idealized" or "Unmeasured".

Note: Assign consecutive numbers to the DUT ports before using this command.

Setting parameters:

<SParamType> TRANsmission | REFLection | XTLK | FEXT | NEXT | BALNs

TRANsmission
All transmission parameters of the DUT (within DUT structures)

REFLection
All reflection parameters of the modeled DUT

XTLK
All crosstalk parameters of the modeled DUT (i.e. transmission parameters between structures)
Requires at least two structures.

FEXT
All far-end crosstalk parameters of the modeled DUT
Requires at least two structures with opposite wiring (`[SENSe:]SNPMeasure:DUT:PART<pos>:WIRE`)

NEXT
All near-end crosstalk parameters of the modeled DUT
Requires at least two structures with the same wiring

BALNs
All S-parameters between the two physical ports on the balanced side of a mixed-mode structure.
Requires at least one mixed-mode structure on the modeled DUT (`[SENSe:]SNPMeasure:DUT:PART:ADD D3|D3R ...`

Usage: Setting only

Options: R&S ZNB-K100

Manual operation: See "Shortcuts" on page 420

[SENSe:]SNPMeasure:DUT:INIT:RANGe <MeasureOrIdealize>, <Begin Responding Port>, <Begin Incident Port>[, <End Responding Port>, <End Incident Port>]

Tells the SNPA to idealize or measure the S-parameters in a "rectangular" range of the modeled DUT's S-matrix.

Setting parameters:

<MeasureOrIdealize> IDEalize | MEASure

IDEalize
Idealize the corresponding parameters, i.e. set their data state to "Idealized".

MEASure
Mark the corresponding parameters for measurement, i.e. set their data state to "Unmeasured".

<Begin Responding Port> First row in the S-matrix

<Begin Incident Port> First column in the S-matrix
$S_{<Begin\ Responding\ Port>,<Begin\ Incident\ Port>}$ is the upper left corner of the selected range.

<End Responding Port> Last row in the S-matrix

<End Incident Port> Last column in the S-matrix
$S_{<End\ Responding\ Port>,<End\ Incident\ Port>}$ is the lower right corner of the selected range.

Example: `:SENSe:SNPMeasure:DUT:INIT:RANGe MEASure, 2,2,4,4`
sets the S-parameters in the range from S_{22} to S_{44} to "Unmeasured".

Usage: Setting only

Options: R&S ZNB-K100

Manual operation: See "Range" on page 419

[SENSe:]SNPMeasure:DUT:NUMBer <NumberingScheme>

Applies an automic numbering to the physical DUT ports. Previously assigned port numbers are overwritten.

For manual assignment of port numbers, see `[SENSe:]SNPMeasure:DUT:PART:ADD` and `[SENSe:]SNPMeasure:DUT:PART<pos>:PORT`.

Setting parameters:
<NumberingScheme> OOL | SEQ

OOL
"Odd on Left"
With L and R denoting the number of physical ports on the left and right side of the DUT, this numbering assigns numbers 2l-1 (l=1,...,L) to the ports on the left, and numbers 2r (r=1,...,R) to the ports on the right (from top to bottom).
Note that "Odd on Left" produces an invalid numbering, if |L-R| > 1.

SEQ
"Sequential"
With L and R denoting the number of (physical) ports on the left and right side of the DUT, this numbering assigns port numbers 1 to L to the ports on the left, and port numbers L+1 to L+R to the ports on the right (from top to bottom).

Example: Reset the SNPA DUT model:
`:SENSe:SNPMeasure:RST`
Add a balanced bidirectional structure with manually defined "Odd on Left" port ordering:
`:SENSe:SNPMeasure:DUT:PART:ADD D4,1,2,3,4`
`SENSe:SNPMeasure:DUT?` returns:
`D4,1,'',2,'',3,'',4,''`
Apply sequential ordering:
`SENSe:SNPMeasure:DUT:NUMBer SEQ`
Then `SENSe:SNPMeasure:DUT?` returns:
`D4,1,'',3,'',2,'',4,''`

Usage: Setting only

Options: R&S ZNB-K100

Manual operation: See "Auto Number Ports" on page 415

[SENSe:]SNPMeasure:DUT:PART<pos>?

Displays the topology of structure `<pos>`.

Returns type of the structure, and for each of the structure's physical ports the user-defined number and label. The ports appear in the order *upper left, upper right, lower left, lower right*.

Suffix:
<pos> Position of the related structure among all structures of the DUT

Example: For the unmodified template "SFP+", `:SENSe1:SNPMeasure:DUT:PART2?` returns
`D4,5,'1A_pos',6,'1B_pos',7,'1A_neg',8,'1B_neg`

Usage: Query only

Options: R&S ZNB-K100

Manual operation: See "DUT topology table" on page 413

[SENSe:]SNPMeasure:DUT:PART:ADD <Part_Type>[, <UL_PortNumber>[, <UR_PortNumber[, <LL_PortNumber>[, <LR_PortNumber>]]]]

Appends a new structure to the DUT model. Its position can be changed using `[SENSe:]SNPMeasure:DUT:PART<pos>:MOVE`.

Specifying port numbers is optional here. However, a consecutive numbering of the DUT ports is required when you want to import S-parameters (`[SENSe:]SNPMeasure:DUT:IMPort`) or initialize the measurement (`[SENSe:]SNPMeasure:DUT:INIT:...`).

Setting parameters:
<Part_Type> D1 | D1R | D2 | D2D | D2DR | D3 | D3R | D4
Structure type (see Table 5-3)

<UL_PortNumber>, ..., <LR_PortNumber> Port numbers, from upper left to lower right

Example: Adding a differential through structure as the first structure of a DUT at the GUI:
`[SENSe:]SNPMeasure:DUT:PART:ADD D4,1,2,3,4`

Usage: Setting only

Options: R&S ZNB-K100

Manual operation: See "Add Structure" on page 412

[SENSe:]SNPMeasure:DUT:PART<pos>:DELete

Deletes DUT structure no. `<pos>`.

Suffix:
<pos> Position of the structure among all structures of the modeled DUT

Usage: Event

Options: R&S ZNB-K100

Manual operation: See "Structure" on page 414

[SENSe:]SNPMeasure:DUT:PART<pos>:MOVE <Direction>

Moves the structure at position <pos> one position up/down in the DUT topology model, without changing the port numbering or other properties of the affected structures.

Suffix:
<pos> Current position of the structure, among all structures of the modeled DUT

Setting parameters:
<Direction> UP | DOWN

Usage: Setting only

Options: R&S ZNB-K100

Manual operation: See "Structure" on page 414

[SENSe:]SNPMeasure:DUT:PART<pos>:PORT <PortPosition>, <PortNumber>[, <PortName>]

Setting: Modifies structure `<pos>` of the SNPA's DUT model at the specified `<PortPosition>` by assigning a new port number (and label).

Query:

- `:SENSe:SNPMeasure:DUT:PART<pos>:PORT? <PortPosition>` returns the port number and label at port position `<PortPosition>` of structure `<pos>`.

- `:SENSe:SNPMeasure:DUT:PART<pos>:PORT?` returns the port number and label at all port positions of structure `<pos>`.

Suffix:
<pos> Position of the related structure among all structures of the DUT

Parameters:
<PortPosition> UPLeft | UPRight | LOLeft | LORight
Upper left to lower right position.

<PortNumber> Port number
An initial port number can be assigned at the time the structure is created using `[SENSe:]SNPMeasure:DUT:PART:ADD`. `[SENSe:]SNPMeasure:DUT:NUMBer` performs an automatic numbering of all DUT ports.

<PortName> Port label

Example: `:SENSe:SNPMeasure:DUT:PART1:PORT UPLeft,1,'upper left label'`
assigns port number 1 and label 'upper left label' to the upper left port of structure 1.
`:SENSe:SNPMeasure:DUT:PART1:PORT? UPLeft`
returns `1,'upper left label'`
`:SENSe:SNPMeasure:DUT:PART1:PORT?`
returns something like `UPLeft=1,'upper left label';UPRight=2,'';LOLeft=3,'';LORight=4,''`

Options: R&S ZNB-K100

Manual operation: See "Port/Label" on page 414

[SENSe:]SNPMeasure:DUT:PART<pos>:WIRE <LeftSide>

Defines or queries the wiring of a through structure.

Suffix:
<pos> Position of the structure, among all structures of the modeled DUT

Parameters:
<LeftSide> EMPTy | RX | TX
Left side wiring.

EMPTy
Undefined

RX
Connected to a receiver

TX
Connected to a transmitter

Specifying the wiring for the left side determines the wiring of the right side (EMPTY↔EMPTY, RX↔TX, TX↔RX). The query returns the wiring at both ends

Example:
```
:SENSe:SNPMeasure:RST
:SENSe:SNPMeasure:DUT:PART:ADD D2,1,2
:SENSe1:SNPMeasure:DUT:PART1:WIRE TX
:SENSe1:SNPMeasure:DUT:PART1:WIRE?
'TX','RX'
```

Options: R&S ZNB-K100

Manual operation: See "Wiring" on page 413

[SENSe:]SNPMeasure:DUT:PART<pos>:WIRE:IDENtity [<DutSide>[, <Identity>]]

Defines an identity for a balanced endpoint of a through structure with defined wiring (`[SENSe:]SNPMeasure:DUT:PART<pos>:WIRE RX | TX`). For other endpoints, the setting is ignored.

When you define a through structure's wiring for the first time, the SNPA assigns empty identities.

Suffix:
<pos> Position of the structure, among all structures of the modeled DUT

Parameters:
<DutSide> LEFT | RIGHt

<Identity> String value. The firmware neither requires nor enforces uniqueness.

Options: R&S ZNB-K100

Manual operation: See "Identity" on page 414

[SENSe:]SNPMeasure:DUT:PART<pos>:WIRE:POLarity [<Polarity>]

Defines the polarity of a balanced endpoint.

Only available for through structure with defined wiring (`[SENSe:]SNPMeasure:DUT:PART<pos>:WIRE RX | TX`). For other endpoints, the setting is ignored.

When you define a through structure's wiring for the first time, the SNPA assumes that the upper/lower (physical) port has positive/negative polarity.

Suffix:
<pos> Position of the structure among all structures of the modeled DUT

Parameters:
<Polarity> NONE | TOPPositive | TOPNegative

NONE means unspecified.

Options: R&S ZNB-K100

Manual operation: See "Pol." on page 414

[SENSe:]SNPMeasure:DUT:PREFix <Left Side>, <Right Side>

The SNPA models each DUT with a left and a right side. This command allows you to define a descriptive text for each side. It is added as a prefix to the port names in the GUI.

Parameters:

<Left Side> Left side label

<Right Side> Right side label

Options: R&S ZNB-K100

Manual operation: See "Left Side Label/Right Side Label" on page 413

[SENSe:]SNPMeasure:DUT:RIMPedance <Port>[, <Real>[, <Imag>]]

Defines the reference impedance of physical DUT port `<Port>` in the SNPA model. The default impedance is 50 Ω.

Parameters:

<Port> Port number

<Real>, <Imag> Real and imaginary part of the reference impedance.

Options: R&S ZNB-K100

Manual operation: See "Reference Impedance" on page 416

[SENSe:]SNPMeasure:DUT:TEMPlate <Template>

The SNPA comes with a set of pre-configured DUT models. These templates pre-populate the DUT topology table, with default structures, wiring, numbering and labeling. You can then modify the model according to your actual setup.

Note that applying a template resets the SNPA project and hence must be the first configuration step.

Parameters:

<Template> Template name
Use the query to list the available models.

Options: R&S ZNB-K100

Manual operation: See "Use Template" on page 415

[SENSe:]SNPMeasure:EXPort <File>

Saves the complete S-matrix to a Touchstone file.

Execute this command once you have completed the SNPA's data collection steps via measurement (`[SENSe:]SNPMeasure:STEP<pos>:MEASure` or import (`[SENSe:]SNPMeasure:STEP<pos>:IMPort`).

Setting parameters:

<File> A string parameter specifying the name and directory of the Touchstone file.
The SNPA chooses the correct file extension according to the number of physical DUT ports.
If no path is specified, the analyzer uses the default directory `C:\Users\Public\Documents\Rohde-Schwarz\Vna\Traces`.

Usage: Setting only

Options: R&S ZNB-K100

Manual operation: See "Export snp File..." on page 412

[SENSe:]SNPMeasure:INSTrument:PORT [<Ports>]...

Defines or queries the ports on the VNA and connected switch matrices that are available for measuring the modeled DUT.

By default, the SNPA uses **all** available ports – disregarding their state in the logical port configuration (recommended).

Parameters:

<Ports> Comma-separated list of port numbers

Options: R&S ZNB-K100

Manual operation: See "VNA" on page 415

[SENSe:]SNPMeasure:MINTegrity

`SENSe:SNPMeasure:MINTegrity?` returns the result of the measurement integrity check:

- `1`, if the active channel's sweep settings match the sweep settings of the current SNPA project.
  In this case, you can proceed with the measurement steps (`[SENSe:]SNPMeasure:STEP<pos>:MEASure`).
- `0`, otherwise.
  In this case, you can use `[SENSe:]SNPMeasure:MINTegrity:RESTore` to adjust the channel's sweep settings to the project's sweep settings.

`SENSe:SNPMeasure:MINTegrity` locks the sweep settings of the current SNPA project. Requires the measurement steps to be initialized (using one of the `[SENSe:]SNPMeasure:DUT:INIT` commands).

Options: R&S ZNB-K100

Manual operation: See "Measurement" on page 430

[SENSe:]SNPMeasure:MINTegrity:RESTore

Restores "measurement integrity", i.e. adjusts the active channel's sweep settings to the sweep settings of the current SNPA project.

Without measurement integrity, `[SENSe:]SNPMeasure:STEP<pos>:MEASure` will fail.

Use `[SENSe:]SNPMeasure:MINTegrity` on page 1464 to query the measurement integrity state.

Usage: Event

Options: R&S ZNB-K100

Manual operation: See "Measurement" on page 430

[SENSe:]SNPMeasure:RECall <File>

Loads an SNPA project from file (`*.snp`).

Note:

- SNPA project files can be created using the command `[SENSe:]SNPMeasure:SAVE`.
- Make sure to save your current project before loading another one.

Setting parameters:

<File> A string parameter specifying the name and directory of the `snp` file.

Usage: Setting only

Options: R&S ZNB-K100

Manual operation: See "Recall..." on page 411

[SENSe:]SNPMeasure:RST

Resets the current SNPA project or creates a new one.

Use `[SENSe:]SNPMeasure:SAVE` to save the current project before using the reset command.

Usage: Event

Options: R&S ZNB-K100

Manual operation: See "New..." on page 411

[SENSe:]SNPMeasure:SAVE <File>

Saves the current SNPA project to file (`*.snp`).

Use the complementary command `[SENSe:]SNPMeasure:RECall` to load a project from file.

Setting parameters:

<File> A string parameter specifying the name and directory of the `snp` file.
If no path is specified, the analyzer uses the default directory `C:\Users\Public\Documents\Rohde-Schwarz\Vna\Traces`.

Usage: Setting only

Options: R&S ZNB-K100

Manual operation: See "Save..." on page 411

[SENSe:]SNPMeasure:STATus?

Queries the execution state of the measurement steps.

Returns a string that looks something like `1:2[DONE];2:2[TODO]`, which means step 1 of 2 is done, while step 2 of 2 is still to do.

Available once you have completed the measurement setup (DUT topology, VNA ports etc.) and if not all S-parameters are "Idealized".

Example: Suppose we're on a **2-port VNA** without switch matrices.

```
// Reset the SNPA:
:SENSe:SNPMeasure:RST
// :SENSe:SNPMeasure:STATus? would give an error
// Add a balanced through structure:
:SENSe:SNPMeasure:DUT:PART1:ADD D4,1,2,3,4
// :SENSe:SNPMeasure:STATus? would still give an error
// Tell the SNPA to measure transmission parameters only:
:SENSe:SNPMeasure:DUT:INIT:MEASure TRANsmission
// Now you can query the execution states:
:SENSe:SNPMeasure:STATus?
// 1:4[TODO];2:4[TODO];3:4[TODO];4:4[TODO]
// Measure step 1 and query the execution states again:
:SENSe:SNPMeasure:STEP1:MEASure
:SENSe:SNPMeasure:STATus?
// 1:4[DONE];2:4[TODO];3:4[TODO];4:4[TODO]
```

Usage: Query only

Options: R&S ZNB-K100

Manual operation: See "Step heading" on page 427

[SENSe:]SNPMeasure:STEP<pos>?

Returns the state of SNPA measurement step `<pos>`, along with its mandated "VNA to DUT Connections" and "DUT Terminations", and the S-parameters to be measured.

`:SENSe:SNPMeasure:STEP1?` returns something like `1:4[TODO];Vp1=Dp1,Vp3=Dp2;X3,X4;S12,S21`, which means:

- Step 1 of 4 is still to do (not already `[DONE]`)
- Before starting the measurement (`[SENSe:]SNPMeasure:STEP<pos>:MEASure`):
  - Connect VNA port 1 to DUT port 1, and VNA port 2 to DUT port 2
  - Germinate DUT ports 3 and 4
- Step 1 measures S-parameters S12 and S21.

Suffix:
<pos> Step number

Example: Suppose we are on a **2-port VNA** without switch matrices.

```
// Reset the SNPA:
:SENSe:SNPMeasure:RST
// Add a balanced through structure:
:SENSe:SNPMeasure:DUT:PART1:ADD D4,1,2,3,4
// Tell the SNPA to measure transmission parameters only:
:SENSe:SNPMeasure:DUT:INIT:MEASure TRANsmission
// Query the execution states:
:SENSe:SNPMeasure:STATus?
// 1:4[TODO];2:4[TODO];3:4[TODO];4:4[TODO]
// Query state, connections and terminations for step 1:
:SENSe:SNPMeasure:STEP1?
// 1:4[TODO];Vp1=Dp1,Vp2=Dp2;X3,X4;S12,S21
```

Usage: Query only

Options: R&S ZNB-K100

Manual operation: See "VNA to DUT Connections and DUT Terminations" on page 427

[SENSe:]SNPMeasure:STEP<pos>:CLEar

Use this command to mark the S-parameters to be measured in step <pos> as "Unmeasured".

You can then decide whether you want to import or measure them (`[SENSe:]SNPMeasure:STEP<pos>:IMPort`, `[SENSe:]SNPMeasure:STEP<pos>:MEASure`).

Suffix:
<pos> Step number

Usage: Event

Options: R&S ZNB-K100

Manual operation: See "Clear Step" on page 429

[SENSe:]SNPMeasure:STEP<pos>:IMPort <Measurement SParam>, <Touchstone SParam>, <Touchstone File>

Imports one of the S-parameters to be measured in SNPA step <pos> from Touchstone file.

Overwrites previously measured and/or imported data (see `[SENSe:]SNPMeasure:STEP<pos>:MEASure` AND `[SENSe:]SNPMeasure:DUT:IMPort`).

Suffix:
<pos> Step number

Setting parameters:
<Measurement SParam> `'S<i><j>'` where `<i>` and `<j>` are DUT port numbers

<Touchstone SParam> `'S<k><l>'` where `<k>` and `<l>` select the S-parameter in the Touchstone file (by position, not by port name!)

<Touchstone File> String parameter specifying the name and directory of the Touchstone file.

Usage: Setting only

Options: R&S ZNB-K100

Manual operation: See "S-parameters: button grid and panel" on page 428

[SENSe:]SNPMeasure:STEP<pos>:MEASure

If step `<pos>` is `TODO` (see `[SENSe:]SNPMeasure:STEP<pos>?`), you can use this command to measure the previously "Unmeasured" S-parameters of this step.

If it is already `DONE`, but you want to re-measure or re-import some of the related S-parameters, use `[SENSe:]SNPMeasure:STEP<pos>:CLEar` on page 1467 to make them "Unmeasured" again.

Suffix:
<pos> Step number

Usage: Event

Options: R&S ZNB-K100

Manual operation: See "Start Sweep" on page 428

7.3.14.18 [SENSe:]SWEep...

The `[SENSe:]SWEep...` commands provide general settings to control the sweep. Most of the settings have an impact on the sweep time.

[SENSe<Ch>:]SWEep:AXIS:FREQuency 1469
[SENSe<Ch>:]SWEep:AXIS:POWer 1470
[SENSe<Ch>:]SWEep:COUNt 1471
[SENSe:]SWEep:COUNt:ALL 1471

[SENSe<Ch>:]SWEep:DETector:TIME 1472
[SENSe<Ch>:]SWEep:DWELl 1472
[SENSe<Ch>:]SWEep:DWELl:IPOint 1473
[SENSe<Ch>:]SWEep:GENeration 1473
[SENSe<Ch>:]SWEep:GENeration:ANALog:CONDition? 1474
[SENSe<Ch>:]SWEep:MODE 1475
[SENSe<Ch>:]SWEep:POINts 1475
[SENSe<Chn>:]SWEep:SRCPort 1476
[SENSe<Ch>:]SWEep:STEP 1476
[SENSe<Ch>:]SWEep:TIME 1477
[SENSe<Ch>:]SWEep:TIME:AUTO 1478
[SENSe<Ch>:]SWEep:TYPE 1479

[SENSe<Ch>:]SWEep:AXIS:FREQuency <Scale>

Selects either the channel base frequency or one of the port frequencies as the stimulus axis in all diagrams of the active recall set. This command has no effect if no port-specific frequencies are defined.

Suffix:

<Ch> Channel number

Parameters:

<Scale> String parameter, one of the following signals:
'Channel Base; Source' - base channel frequency
'Port 1; Source' - source frequency at port 1
'Port 2; Source' - source frequency at port 2
...
'Gen 1; Source' - external generator frequency 1
...
'Pmtr 1; Receiver' - external power meter frequency 1
...
*RST: 'Channel Base: Source'

Example:

```
*RST; FREQ:STAR 1 GHz; STOP 2 GHz
```
Define a base channel frequency range between 1 GHz and 2 GHz. This frequency is also used as an (initial) frequency range for all test ports and external generators.
```
SOUR:FREQ1:CONV:ARB:IFR 1, 1, 1E+9, SWE
```
Convert the source frequency at test port no. 1 to the range between 2 GHz and 3 GHz.
```
SWE:AXIS:FREQ 'Port 1; Source'
```
Select the source signal at port 1 as the reference signal for frequency definitions.
```
FREQ:STAR 3 GHz; STOP 4 GHz
```
Shift all frequency ranges by +1 GHz.
```
SWE:AXIS:FREQ 'Port 2; Source'
```
Select the source signal at port 2 as the reference signal for frequency definitions.
```
FREQ:STAR?; STOP?
```
Query the frequency range at test port 2. The response is `2000000000;3000000000`.

Manual operation: See "Stimulus Axis – Frequency / Power" on page 528

[SENSe<Ch>:]SWEep:AXIS:POWer <Scale>

Selects either the channel base power or one of the port powers as the stimulus axis in all diagrams of the active recall set. This command has no effect if no port-specific powers are defined.

Suffix:

<Ch> Channel number

Parameters:

<Scale> String parameter, one of the following signals:
'Channel Base; Source' - base channel power
'Port *p*; Source' - source power at port *p*
'Gen *g*; Source' - source power at external generator *g*

*RST: 'Channel Base; Source'

Example:
```
*RST; SOUR:POW 5
```
Define a base channel power of +5 dBm. This power is also used as an (initial) power for all test ports and external generators.
```
SOUR:POW1:OFFS -5, CPAD
```
Change the source power at test port no. 1 to 0 dB without affecting the power at the remaining ports.
```
SWE:AXIS:POW 'Port 1; Source'
```
Select the source signal at port 1 as the reference signal for power definitions.
```
SOUR:POW -5
```
Shift all powers by -5 GHz.
```
SWE:AXIS:POW 'Port 2; Source'
```
Select the source signal at port 2 as the reference signal for power definitions.
```
SOUR:POW?
```
Query the power at test port 2. The response is 0.

Manual operation: See "Stimulus Axis – Frequency / Power" on page 528

[SENSe<Ch>:]SWEep:COUNt <Sweep>

Defines the channel-specific number of sweeps to be measured and buffered in single sweep mode (`INITiate<Ch>:CONTinuous OFF`). Use `[SENSe:]SWEep:COUNt:ALL` to define the sweep count for all channels.

Suffix:
<Ch> Channel number

Parameters:
<Sweep> Number of consecutive sweeps to be measured and buffered.

Range: 1 to 100000
*RST: 1

Example: See `CALCulate<Chn>:DATA:NSWeep:FIRSt?`

Manual operation: See "Sweeps" on page 559

[SENSe:]SWEep:COUNt:ALL <Sweep>

Defines the number of sweeps to be measured in single sweep mode (`INITiate<Ch>:CONTinuous OFF`). The setting is applied to all channels. Use `[SENSe<Ch>:]SWEep:COUNt` to define the sweep count for a single channel.

Setting parameters:
<Sweep> Number of consecutive sweeps to be measured.

Range: 1 to 100000
*RST: 1

Example: See Section 8.2.4, "Data handling", on page 1668.

Usage: Setting only

Manual operation: See "Sweeps" on page 559

[SENSe<Ch>:]SWEep:DETector:TIME <DetectorTime>

Defines the observation time per sweep point if a detector other than the "Normal" (SAMple) detector is used. The detector is selected together with the measured quantity (`CALCulate<Ch>:PARameter:MEASure` or `CALCulate<Ch>:PARameter:SDEFine`).

Suffix:
<Ch> Channel number

Parameters:
<DetectorTime> Detector time

Range: 0 s to 3456000 s
*RST: 0.01 s
Default unit: s

Example:
```
*RST; :CALC:PAR:MEAS 'TRC1', 'a1D1AVG'
```
Select the wave quantity a_1 for the default trace and activate the AVG detector.
```
SWE:DET:TIME 1
```
Specify an observation time of 1 s at each sweep point.

Manual operation: See "Detector" on page 305

[SENSe<Ch>:]SWEep:DWELl <MeasDelay>

Defines the "Meas. Delay" before each partial measurement or the first partial measurement (see `[SENSe<Ch>:]SWEep:DWELl:IPOint`). Setting a delay disables the automatic calculation of the (minimum) sweep time (see `[SENSe<Ch>:]SWEep:TIME:AUTO`).

Suffix:
<Ch> Channel number

Parameters:
<MeasDelay> Measurement delay before each partial measurement. Changing the delay leaves the number of points unchanged but has an impact on the duration of the sweep (`[SENSe<Ch>:]SWEep:TIME`).

Range: 0 s to 13680 s
*RST: 0 s
Default unit: s

Example:
```
*RST
```
Reset the instrument, activating a frequency sweep with the S-parameter S_{21} as a measurement result for channel and trace no. 1.
```
SWEep:TIME?
```
Query total sweep time.
```
SWEep:DWEL 1
SWEep:DWEL:IPOint ALL
```
Set a delay of 1 s for each partial measurement.
```
SWE:TIME?
```
Query total sweep time. The time is extended by the delay times the total number of sweep points (one partial measurement per sweep point required).

Manual operation: See "Meas Delay" on page 538

[SENSe<Ch>:]SWEep:DWELl:IPOint <InsertionPoints>

Defines whether the measurement delay (previously defined via `[SENSe<Ch>:]SWEep:DWELl`) is inserted before all partial measurements or before the first partial measurement only.

Suffix:
<Ch> Channel number

Parameters:
<InsertionPoints> ALL | FIRSt

Insertion before all or before the first partial measurement

*RST: ALL

Example: See `[SENSe<Ch>:]SWEep:DWELl`

Manual operation: See "All Partial Meas'ments / First Partial Meas'ment" on page 538

[SENSe<Ch>:]SWEep:GENeration <Mode>

On older R&S ZNB, this command defines the sweep mode (stepped or swept) for linear frequency sweeps.

For segmented sweeps, this setting can be overridden by segment-specific sweep modes (`[SENSe<Ch>:]SEGMent<Seg>:SWEep:GENeration`).

Note that swept mode is **not** supported on the following instruments:

- R&S ZNB43
- R&S ZNBT equipped with synth boards SYNTH13DDS_DS or SYNTH13DDS_L, either ex factory or via upgrade.

  These synth boards are shipped with:
  - R&S ZNBT8|26|40 with serial number ≥ 200000
  - R&S ZNBT20 variant 64 (order no. 1332.9002K64)

Use `SYSTem:DFPRint?` to query hardware-related information such as the synth boards equipping of your instrument.

Suffix:
<Ch> Channel number

Parameters:
<Mode> STEPped | ANALog

STEPped
Stepped mode

ANALog
Swept mode

Manual operation: See "Freq Sweep Mode" on page 539

[SENSe<Ch>:]SWEep:GENeration:ANALog:CONDition?

Returns the state of the swept mode (see `[SENSe<Ch>:]SWEep:GENeration` and `[SENSe<Ch>:]SEGMent<Seg>:SWEep:GENeration`), which can be one of the following:

- 0, "Swept mode not active."
- 1, "Swept mode active."
  (for at least one segment in segmented sweeps)
- -1, "Swept mode not possible while external powermeter is used."
- -2, "Swept mode not possible while external generator is used."
- -3, "Swept mode not possible while Point or Partial Measurement Trigger is used."
- -4, "Swept mode not possible while Point Delay is used."
- -5, "Swept mode not possible while Chopped Driving Mode is used."
- -6, "Swept mode not possible while DC Measurement is used."
- -7, "Swept mode not possible while AVG Detector is used."
- -8, "Swept mode not possible: Lin Frequency Sweep only supported!"
- -9, "Swept mode not possible while Arbitrary or Intermodulation Measurement is used."
- -10,"Swept mode not possible: this combination of features is not yet supported!"
- -11, "Swept mode not supported. Please contact Rohde&Schwarz Service for an upgrade."
  This may occur for some "old" synthesizer hardware.

Suffix:
<Ch> Channel number

Return values:
<Code> Status code
Negative values indicate that swept mode could not be activated for the reason described in the <state_desc>.

<Descr> State description

Usage: Query only

Manual operation: See "Freq Sweep Mode" on page 539

[SENSe<Ch>:]SWEep:MODE <arg0>

Sets or queries the sweep mode of channel <Ch>.

Suffix:
<Ch> Channel number

Parameters:
<arg0>
CONTinuous
Continuous sweep mode

SINGle
Single sweep mode, sweeping

HOLD
Single sweep mode, not sweeping

Manual operation: See "Continuous / Single" on page 559

[SENSe<Ch>:]SWEep:POINts <SweepPoint>

Defines the total number of measurement points per sweep ("Number of Points").

Values between 1 and 100001 can be set.

Suffix:
<Ch> Channel number 1

Parameters:
<SweepPoint> Number of points per sweep, in the range of 1 and 100001 for frequency sweeps.

Range: see above
*RST: 201

Example:
```
*RST
```
Reset the instrument, activating a frequency sweep with 201 sweep points.
```
SWE:TIME?
```
Query total sweep time.
```
SWE:POIN 2010
```
Multiply the (default) number of points by 10.
```
SWE:TIME?
```
Query total sweep time again. The analyzer estimates a sweep time that is also multiplied by 10.

Manual operation: See "Number of Points" on page 527

[SENSe<Chn>:]SWEep:SRCPort <Port>

Sets/gets the source port for the stimulus signal. The setting acts on the active trace. The effect of the drive port selection depends on the measurement parameter associated to the active trace:

- If an S-parameter $S_{<out><in>}$ is measured, the second port number index <in> (input port of the DUT = drive port of the analyzer) is set equal to the selected drive port: Drive port selection affects the measured quantity.
- If a wave quantity or a ratio is measured, the drive port is independent of the measured quantity:

Suffix:
<Chn> Channel number used to identify the active trace

Parameters:
<Port> Logical port number
*RST: 1

Example:
```
CALC4:PAR:SDEF "Ch4Tr1", "A1"
```
Create channel 4 and a trace named "Ch4Tr1" to measure the wave quantity a_1. The trace automatically becomes the active trace.
```
SENS4:SWE:SRCP 2
```
Select drive port 2 for the active trace.
```
CALC4:PAR:MEAS? "Ch4Tr1"
```
Query the measurement result for "Ch4Tr1". The response is `'A1D2SAM'`.

[SENSe<Ch>:]SWEep:STEP <StepSize>

Sets the distance between two consecutive sweep points in a (non-segmented) linear frequency sweep.

Suffix:
<Ch> Channel number

Parameters:

<StepSize> Frequency step size.
The step size is equal to the current sweep spandivided by the number of sweep points minus one (see `[SENSe<Ch>:]FREQuency:SPAN?` and `[SENSe<Ch>:]SWEep:POINts?`, respectively). See also the description of manual control and the program example below.

Range: Depends on the current sweep span and the maximum number of sweep points: "Span Frequency" / (max. "Number of Points" – 1) ≤ "Freq Step Size" ≤ "Span Frequency"

*RST: Depends on the analyzer model. The default step size is equal to the default sweep span of the analyzer divided by the default number of sweep points minus one.

Default unit: Hz

Example:

```
*RST; :SWE:STEP?
```
Query the default step size. Currently for all analyzers the default sweep span is the full frequency range and the default number of points is 201. Hence the response is (max. frequency - min. frequency) / 200.

```
SWE:STEP UP
```
Increase the step size.

```
FREQ:STOP?; :SWE:POIN?
```
Query the stop frequency of the sweep and the number of points. Increasing the step size has changed both values.

Manual operation: See "Freq Step Size" on page 537

[SENSe<Ch>:]SWEep:TIME <SweepDuration>

Sets the duration of the sweep ("Sweep Time"). Setting a sweep time disables the automatic calculation of the (minimum) sweep time; see `[SENSe<Ch>:]SWEep:TIME:AUTO`.

Note: The sweep time is ignored for the sweep types "Time" and "CW Mode" (`[SENSe<Ch>:]SWEep:TYPE`).

Suffix:

<Ch> Channel number

Parameters:

<SweepDuration> Sweep time. The minimum possible sweep time depends on the other channel settings, in particular on the number of points (`[SENSe<Ch>:]SWEep:POINts`), the IF bandwidth (`[SENSe<Ch>:]BWIDth[:RESolution]`) and the measurement delay for each partial measurement (`[SENSe<Ch>:]SWEep:DWELl`). Changing the duration leaves the number of points unchanged but directly affects the delay.

Range: Between minimum sweep time (for current channel settings) and 100000 s
*RST: Minimum sweep time (for *RST channel settings)
Default unit: s

Example:

```
SWE:TIME 1
```
Set a total sweep time of 1 s.
```
SWE:DWEL?
```
Query the delay for each partial measurement.
```
SWE:TIME 2
```
Increase the total sweep time to 2 s.
```
SWE:DWEL?
```
Query the meas. delay for each partial measurement again. The delay is increased by 1 s divided by the total number of partial measurements per sweep.

Manual operation: See "Stop Time" on page 528

[SENSe<Ch>:]SWEep:TIME:AUTO <Boolean>

When enabled, the (minimum) sweep time is calculated internally using the other channel settings and zero delay (`[SENSe<Ch>:]SWEep:DWELl`).

Note: The automatically calculated sweep duration is ignored for the sweep types "Time" and "CW Mode" (`[SENSe<Ch>:]SWEep:TYPE`).

Suffix:

<Ch> Channel number

Parameters:

<Boolean> ON | OFF - turn the automatic calculation of the sweep time on or off.
OFF is also set if the sweep duration or delay is set explicitly using `[SENSe<Ch>:]SWEep:TIME` or `[SENSe<Ch>:]SWEep:DWELl`.

*RST: ON

Example:

```
SWE:TIME 1
```
Set a total sweep time of 1 s.
```
SWE:TIME:AUTO?
```
A query returns the value `1`.

Manual operation: See "Sweep Time / Auto" on page 538

[SENSe<Ch>:]SWEep:TYPE <Format>

Selects the sweep type, i.e. the sweep variable (frequency/power/time) and the position of the sweep points across the sweep range.

Suffix:
<Ch> Channel number.

Parameters:
<Format> LINear | LOGarithmic | POWer | CW | POINt | SEGMent

LINear
Linear frequency sweep at constant source power (SOURce<Ch>:POWer<PhyPt>[:LEVel][:IMMediate][:AMPLitude]). The stimulus frequency ([SENSe<Ch>:]FREQuency:...) is swept in equidistant steps over the frequency range. In a Cartesian diagram, the x-axis is a linear frequency axis.

LOGarithmic
Logarithmic frequency sweep. The frequency is swept in equidistant steps on a logarithmic scale. In a Cartesian diagram, the x-axis is a logarithmic frequency axis.

POWer
Power sweep. The measurement is performed at constant frequency (SOURce<Ch>:FREQuency<PhyPt>:FIXed) but with variable generator power that is swept in linear, equidistant steps over a continuous range (SOURce<Ch>:POWer<PhyPt>:STARt, SOURce<Ch>:POWer<PhyPt>:STOP). In a Cartesian diagram, the x-axis is a dB-linear power axis.

CW
Time sweep. The measurement is performed at constant frequency (SOURce<Ch>:FREQuency<PhyPt>:FIXed) and source power (SOURce<Ch>:POWer<PhyPt>[:LEVel][:IMMediate][:AMPLitude]) and repeated over a specified period of time at constant time intervals.

POINt
CW mode sweep, time sweep triggered according to the current trigger settings.
SEGMent - segmented frequency sweep. The sweep range is composed of several continuous frequency ranges or single frequency points defined by means of the commands in the [SENSe<Ch>:]SEGMent<Seg>... subsystem.

*RST: LINear

Example:
```
*RST
```
Reset the analyzer, activating a linear frequency sweep.
```
SWE:TYPE LOG
```
Change to a logarithmic frequency sweep, resetting the stimulus values of the sweep points.

Manual operation: See "Lin Freq" on page 540

7.3.14.19 [SENSe:]... (other)

[SENSe<Ch>:]COUPle 1480
[SENSe<Ch>:]DC<DCInp>:RANGe 1480
[SENSe<Ch>:]PORT<PhyPt>:ZREFerence 1481
[SENSe:]UDSParams:ACTive 1482
[SENSe:]UDSParams<Pt>:PARam 1482

[SENSe<Ch>:]COUPle <Mode>

Determines the order of partial measurements and sweeps (see Section 4.1.4.1, "Partial measurements and driving mode", on page 83).

Suffix:
<Ch> Channel number. This suffix is ignored; the sweep mode applies to all channels in the active recall set.

Parameters:
<Mode> AUTO | ALL | NONE

AUTO
Automatic mode: fast sweeps are performed in alternated driving mode, slower sweeps in chopped mode

ALL
Chopped driving mode, complete all partial measurements before proceeding to the next sweep point

NONE
Alternated driving mode on, reverse the order of partial measurements and sweeps

*RST: AUTO
Note: The "Block Alternated" mode (`BLOCk`) and the related optional parameter `<NumberOfAlternatedBlocks>` were removed with FW V4.0. For compatibility reasons, `BLOCk` is mapped to `NONE` and `<NumberOfAlternatedBlocks>` is simply ignored.

Example:
```
COUP NONE
```
Activate the alternated driving mode.
```
TRIG:LINK 'PPO'
```
Set the triggered measurement sequence equal to one partial measurement. Each trigger event starts one partial measurement for all sweep points.

Manual operation: See "Driving Mode" on page 699

[SENSe<Ch>:]DC<DCInp>:RANGe <Voltage>

Assigns a DC voltage range to the rear panel input connector DC INPUT <DCInp>.

Suffix:

<Ch> Channel number

<DCInp> Number of DC INPUT connector on the rear panel, 1 to 4

Parameters:

<Voltage> DC voltage range. The entered numeric value is rounded up to ±300 mV, ±3 V, ±20 V. Values above +20 V are rounded down.

Default unit: V

Example: `SENSe1:DC2:RANGe 1; :DC2:RANGE?`
Select a DC measurement at DC INPUT2 with an expected DC voltage range of ±1 V. The value is rounded up; the response is `3`.

Manual operation: See "Ranges" on page 320

[SENSe<Ch>:]PORT<PhyPt>:ZREFerence <RealPart>[, <ImaginaryPart>]

Specifies the complex reference impedance for the physical port numbered `<PhyPt>` (impedance renormalization).

Use `[SENSe<Ch>:]LPORt<LogPt>:ZDEFault[:STATe]` to toggle between configured and default reference impedances.

Suffix:

<Ch> Channel number

<PhyPt> Physical port number

Parameters:

<RealPart> Real part of the port impedance.

Range: 1 mΩ to 10 MΩ
*RST: Default reference impedance of the connector family assigned to the port (real impedance, e.g. 50 Ω).
Default unit: Ohm

<ImaginaryPart> Imaginary part of the port impedance. This part may be omitted to define a real impedance.

Range: - 10 MΩ to 10 MΩ
*RST: Default reference impedance of the connector family assigned to the port (real impedance, e.g. 50 Ω).
Default unit: Ohm

Example: `PORT2:ZREF 52, 2`
Specify a complex reference impedance of 52 Ω + j * 2 Ω for the (physical) port no. 2.
`CALC:TRAN:IMP:RNOR PWAV`
Select renormalization of port impedances according to the power waves theory.

Manual operation: See "Single Ended Mode / Common Mode / Differential Mode" on page 302

[SENSe:]UDSParams:ACTive <arg0>

With a "set to OFF", this command disables user-defined physical ports (see Section 4.3.1.2, "Redefined S-parameters", on page 118). In "get direction" it queries whether or not user-defined phasical ports are configured.

Note that a "set to ON" doesn't have any effect; user defined ports can ony be created using `[SENSe:]UDSParams<Pt>:PARam` on page 1482

Parameters:

<arg0> OFF (0) or ON (1)

Example: see `[SENSe:]UDSParams<Pt>:PARam`

Manual operation: See "a wave, b wave, Source" on page 865

[SENSe:]UDSParams<Pt>:PARam <PortString>

Creates or redefines a user-defined physical port (see Section 4.3.1.2, "Redefined S-parameters", on page 118). Causes a factory reset.

Note that the factory reset deletes all switch matrix RF connections, so the RF configuration for switch matrices (see `SYSTem:COMMunicate:RDEVice:SMATrix:CONFigure:STARt`) has to be done *after* the port redefinition.

Suffix:

<Pt> Number of the redefined physical port.

Parameters:

<PortString> String representation '<Source>:<a wave>:<b wave>' where
<Source> = *i* refers to (original) VNA ports
<Source> = G*i* refers to external generator ports
<a wave>, <b wave> is one of the original a*i* or b*i*,
without reusing the same port in different (redefined) ports,
Note that an external generator must be configured before it can be used as <Source>.

Example: `SENSe:UDSParams:ACTive OFF`
resolves all redefined ports
`SENSe:UDSParams1:PARam '1:b2:b3'`
redefines physical port 1 as source = (original) physical port 1, reference receiver = physical port 2, measurement receiver = physical port 3
`SENSe:UDSParams:ACTive?`
returns 1 (= ON; see `[SENSe:]UDSParams:ACTive`)

Manual operation: See "a wave, b wave, Source" on page 865

7.3.15 SOURce commands

The `SOURce...` commands affect the source settings of the R&S ZNB.

Port-specific and general settings

The SOURce... subsystem comprises channel-specific and general settings. Channel-specific settings are valid for the channel specified by the numeric suffix <Ch> (SENSe<Ch>:...). General settings are valid for all channels; the channel suffix is ignored. Refer to the description of the individual commands for more information.

7.3.15.1 SOURce:FREQuency...

The SOURce:FREQuency... commands configure the sources for frequency conversion measurements and control the frequency and power of the internal signal source.

SOURce<Ch>:FREQuency<PhyPt>:CONVersion:ARBitrary:EFRequency<Gen> 1483
SOURce<Ch>:FREQuency<PhyPt>:CONVersion:ARBitrary:IFRequency 1484
SOURce<Ch>:FREQuency:CONVersion:MIXer:PMFixed 1485
SOURce<Ch>:FREQuency:CONVersion:MIXer:PMODe 1486
SOURce<Ch>:FREQuency<PhyPt>[:CW] 1487
SOURce<Ch>:FREQuency<PhyPt>:FIXed 1487

SOURce<Ch>:FREQuency<PhyPt>:CONVersion:ARBitrary:EFRequency<Gen>
<Boolean>, <BaseRatNum>, <BaseRatDen>, <Offset>, <SweepType>[, <OffsRatNum>[, <OffsRatDen>]]

Defines an external generator frequency for frequency-converting measurements. The external generator frequency is either a range (for frequency sweeps) or a CW frequency (for power, time and CW Mode sweeps).

Suffix:

<Ch> Channel number

<PhyPt> Test port number of the analyzer. This suffix is ignored because the generator is referenced via <Gen>.

<Gen> Generator number

Parameters:

<Boolean> Switch the generator on or off.

<BaseRatNum>, <BaseRatDen> Base ratio numerator and denominator.
The specified numerator is rounded to the next higher integer; zero is not allowed. The specified denominator must be positive. It is rounded to the next higher integer.

<Offset> See above, <Numerator>

Range: The converted frequency or frequency range must be within the external generator's frequency range. In addition the <Offset> values are rounded to (positive or negative) multiples of 1 Hz.
*RST: 0
Default unit: Hz

<SweepType> CW | FIXed | SWEep

SWEep - the full formula is applied. For frequency sweeps, the command defines a sweep range.
CW | FIXed - the reduced formula with $f_b = 0$ is applied; the command defines a fixed frequency.

*RST: SWEep

<OffsRatNum>, <OffsRatDen> Offset ratio numerator and denominator.
The specified numerator is rounded to the next higher integer; zero is not allowed. The specified denominator must be positive. It is rounded to the next higher integer.

Example:
```
*RST; FREQ:STAR 1E+9; STOP 1.1E+9
```
Reset the analyzer (activating a frequency sweep) and set the sweep range between 1 GHz and 1.1 GHz.
```
SYST:COMM:RDEV:GEN1:DEF 'Ext. Gen. 1', 'SME02', 'gpib0', '21'
```
Configure an R&S SME02 generator as external generator no. 1, assigning the name "Ext. Gen. 1" and a GPIB address 21.
```
SOUR:FREQ:CONV:ARB:EFR ON, 1, 4, 1E+9, SWE
```
Switch the generator on and convert the generator frequency to the range between 1.25 GHz and 1.275 GHz.

Manual operation: See "Frequency Conversion Formula" on page 689

SOURce<Ch>:FREQuency<PhyPt>:CONVersion:ARBitrary:IFRequency <BaseRatNum>, <BaseRatDen>, <Offset>, <SweepType>[, <OffsRatNum>[, <OffsRatDen>]]

Defines the port-specific frequency for frequency-converting measurements.

The port frequency f_p is calculated according to

f_p = <BaseRatNum>/<BaseRatDen> * f_b + <OffsRatNum>/<OffsRatDen> * <Offset>,

where f_b represents the channel base frequency (parameter `SWEep`). For parameters `CW` or `FIXed`, $f_b = 0$.

Note: The frequency formula is applied even if the analyzer returns an error message, because the frequency is outside the allowed range.

Suffix:

<Ch> Channel number

<PhyPt> Test port number of the analyzer

Parameters:

<BaseRatNum>, <BaseRatDen> Base ratio numerator and denominator.
The specified numerator is rounded to the next higher integer; zero is not allowed. The specified denominator must be positive. It is rounded to the next higher integer.

<Offset> Offset value, rounded (integer)

*RST: 0
Default unit: Hz

<SweepType> CW | FIXed | SWEep

SWEep - the full formula is applied. For frequency sweeps, the command defines a sweep range.
CW | FIXed - the reduced formula with $f_b = 0$ is applied; the command defines a fixed frequency.

*RST: SWEep

<OffsRatNum>, <OffsRatDen> Offset ratio numerator and denominator.
The specified numerator is rounded to the next higher integer; zero is not allowed. The specified denominator must be positive. It is rounded to the next higher integer.

Example:
```
*RST; FREQ:STAR 1E+9; STOP 1.1E+9
```
Reset the analyzer (activating a frequency sweep) and set the sweep range between 1 GHz and 1.1 GHz.
```
SOUR:FREQ2:CONV:ARB:IFR  2, 1, 1E+9, SWE
```
Convert the source frequency at test port no. 2 to the range between 3 GHz and 3.2 GHz.
```
SENSe:FREQuency:CONVersion:GAIN:LMCorrection ON
```
Enable the load match correction for all conversion gains.
```
SENSe:FREQuency:CONVersion?
```
Query the measurement mode. The response is `ARB`, because the port frequency definition also activates the arbitrary mode.
```
SENSe:FREQuency:CONVersion FUND
```
De-activate the arbitrary mode, cancel the port frequency definition.

Manual operation: See "Source Freq. Conversion" on page 688

SOURce<Ch>:FREQuency:CONVersion:MIXer:PMFixed <PortRole>[, <Power>]

Defines a fixed power for the ports involved in a mixer measurements.

Only takes effect if `SOURce<Ch>:FREQuency:CONVersion:MIXer:PMODe` is set to `FIXed` for the related port.

Suffix:
<Ch> Channel number

Parameters:
<PortRole> RF | LO | LO1 | LO2 | IF

See `SOURce<Ch>:FREQuency:CONVersion:MIXer:PMODe` on page 1486.

<Power> Fixed power value

Range: The exact range depends on the analyzer model or external generator used; refer to the data sheet.
*RST: -25 dBm (for LO)
Default unit: dBm

Example: See `[SENSe<Ch>:]FREQuency:CONVersion:MIXer:MFFixed`

Options: R&S ZNB-K4

Manual operation: See "Power tab" on page 329

SOURce<Ch>:FREQuency:CONVersion:MIXer:PMODe <PortRole>[, <PowerMode>]

Sets the power mode for the ports involved in a mixer measurements.

Suffix:
<Ch> Channel number

Parameters:
<PortRole> RF | LO | LO1 | LO2 | IF

RF
Mixer input signal

LO | LO1
Local oscillator signal no. 1

LO2
Local oscillator signal no. 2, for 2-stage mixer measurements (`[SENSe<Ch>:]FREQuency:CONVersion:MIXer:STAGes`)

IF
Mixer IF output signal (of the 2nd mixer, if avalable)

<PowerMode> FUNDamental | FIXed

FIXed – use a fixed power, to be specified via `SOURce<Ch>:FREQuency:CONVersion:MIXer:PMFixed`
FUNDamental – use the channel base power specified via `SOURce<Ch>:POWer<Pt>:STARt, SOURce<Ch>:POWer<Pt>:STOP, SOURce<Ch>:POWer<Pt>[:LEVel][:IMMediate][:AMPlitude]`

*RST: Minimum of the analyzer's frequency range

Example: See `[SENSe<Ch>:]FREQuency:CONVersion:MIXer:MFFixed`

Options: R&S ZNB-K4

Manual operation: See "Power tab" on page 329

SOURce<Ch>:FREQuency<PhyPt>[:CW] <FixedFreq>
SOURce<Ch>:FREQuency<PhyPt>:FIXed <FixedFreq>

Defines the fixed (Continuous Wave, CW) frequency for all sweep types operating at fixed frequency ("Power", "Time", "CW Mode"). The two command forms `SOURce<Ch>:FREQuency<PhyPt>:CW` and `SOURce<Ch>:FREQuency<PhyPt>:FIXed` are equivalent.

The frequency range depends on the R&S ZNB model; see Table 7-18.

Note: `SOURce<Ch>:FREQuency<PhyPt>[:CW]|:FIXed` is equivalent to `[SENSe<Ch>:]FREQuency[:CW]|:FIXed`. Source and receiver frequency are always equal; the four commands overwrite each other.

With option R&S ZNB-K4, Frequency Conversion Measurements, port-specific source and receiver frequencies can be defined; see `SOURce<Ch>:FREQuency<PhyPt>:CONVersion:ARBitrary:IFRequency`.

Suffix:

<Ch> Channel number

<PhyPt> Test port number of the analyzer. This suffix is ignored because the selected frequency applies to all source ports in the active channel.

Parameters:

<FixedFreq> Fixed stimulus and analyzer frequency.

*RST: 1 GHz
Default unit: Hz

Example: `FUNC "XTIMe:POW:A1"`
Activate a time sweep and select the wave quantity a_1 as measured parameter for channel and trace no. 1.
`FREQ:CW 100MHz`
Set the CW frequency to 100 MHz.

Manual operation: See "CW Frequency" on page 528

7.3.15.2 SOURce:GROup...

The `SOURce:GROup...` commands allow you to configure port groups and DUTs. You can define:

- One port group (the group of active logical ports)
- Two or more port groups (parallel measurement of multiple DUTs)

SOURce<Ch>:GROup<Grp> 1488
SOURce<Ch>:GROup<Grp>:CLEar 1488
SOURce<Ch>:GROup:COUNt? 1489
SOURce<Ch>:GROup<Grp>:DPORt:COUNt 1489
SOURce<Ch>:GROup<Grp>:NAME 1489
SOURce<Ch>:GROup<Grp>:PORTs 1490
SOURce<Ch>:GROup<Grp>:PPORt<PhyPort>:DPORt 1490
SOURce<Ch>:GROup<Grp>:PPORts 1491

SOURce<Ch>:GROup<Grp>:SIMultaneous:FOFFset:CONDition....1491
SOURce<Ch>:GROup<Grp>:SIMultaneous:FOFFset:MOFFset:BWFactor....1493
SOURce<Ch>:GROup<Grp>:SIMultaneous:FOFFset:MOFFset:DVALue....1493
SOURce<Ch>:GROup<Grp>:SIMultaneous:FOFFset:MOFFset:MODE....1494
SOURce<Ch>:GROup<Grp>:SIMultaneous:FOFFset:STATe....1495

SOURce<Ch>:GROup<Grp> <first_lp>, <last_lp>

Setting: defines the set of active logical ports (the "master group") as the continuous range from `<first_lp>` to `<last_lp>`, i.e. logical ports outside this range are disabled.

Query: returns the minimum and maximum enabled logical port of group <Grp>.

See `SOURce<Ch>:LPORt<LogPt>` on how to define logical ports.

Note that setting `SOURce<Ch>:GROup` or `SOURce<Ch>:GROup<Grp>:PORTs` automatically dissolves the port groups created using `SOURce<Ch>:GROup<Grp>:PPORts`.

Suffix:

<Ch> Channel number.

<Grp> Port group number.
In "set direction", <Grp> must be 1 (or omitted). In "get direction" also port groups created using `SOURce<Ch>:GROup<Grp>:PPORts` can be queried.

Parameters:

<first_lp>, <last_lp> First and last logical port number in the port group. Must be omitted if the command is used as a query.

Example: See `SOURce<Ch>:LPORt<LogPt>` on page 1531

SOURce<Ch>:GROup<Grp>:CLEar <Scope>

Dissolves port group `<Grp>` or all port groups created using `SOURce<Ch>:GROup<Grp>:PPORts`.

Suffix:

<Ch> Channel number.

<Grp> Port group number.

Setting parameters:

<Scope> ALL

If `ALL` is specified, all port groups are dissolved and a default port group 1, consisting of all available ports, is restored; the `<Grp>` suffix is ignored.
If `ALL` is omitted, only the specified port group is dissolved, except the deleted port group was the only one, in which case again default port group 1 is restored. In case an undefined port group number is used, the analyzer generates an error message.

Example: See `SOURce<Ch>:LPORt<LogPt>` on page 1531

Usage: Setting only

SOURce<Ch>:GROup:COUNt?

Queries the number of port groups in channel no. `<Ch>`.

Suffix:

<Ch> Channel number.

Example: See `SOURce<Ch>:GROup<Grp>` on page 1488

Usage: Query only

Manual operation: See "Controls and Functions" on page 695

SOURce<Ch>:GROup<Grp>:DPORt:COUNt <NumPorts>

Sets/gets the number of ports on DUT `<Grp>`, created using `SOURce<Ch>:GROup<Grp>:PPORts`.

The connections between physical VNA ports and DUT ports are defined using `SOURce<Ch>:GROup<Grp>:PPORt<PhyPort>:DPORt`.

Suffix:

<Ch> Channel number

<Grp> Port group number

Parameters:

<NumPorts> Number of connected ports
Can be higher than the number of ports in port group <Grp>, but not lower.

Example: See `SOURce<Ch>:GROup<Grp>:PPORt<PhyPort>:DPORt`

Manual operation: See "Controls and Functions" on page 695

SOURce<Ch>:GROup<Grp>:NAME <GroupName>

Sets/gets the name of port group `<Grp>`.

Port groups can be created using SOURce<Ch>:GROup<Grp>:PPORts.

Suffix:
<Ch> Channel number

<Grp> Port group number

Parameters:
<GroupName> Group name

Example: See SOURce<Ch>:GROup<Grp>:PPORt<PhyPort>:DPORt

Manual operation: See "Controls and Functions" on page 695

SOURce<Ch>:GROup<Grp>:PORTs <lport0>[, <lport1>[, ...]]

Setting: Defines the set of active logical ports (the "master group") as an arbitrary selection of logical ports. In contrast to SOURce<Ch>:GROup<Grp>, the active logical ports do not have to be numbered consecutively.

Query: Returns the logical ports in the respective group (for numbers > 1 created using SOURce<Ch>:GROup<Grp>:PPORts).

Note that setting SOURce<Ch>:GROup<Grp> or SOURce<Ch>:GROup<Grp>:PORTs dissolves the port groups created using SOURce<Ch>:GROup<Grp>:PPORts (R&S ZNBT only).

See SOURce<Ch>:LPORt<LogPt> on how to define logical ports.

Suffix:
<Ch> Channel number.

<Grp> Port group number.
Setting: <Grp> must be 1 (or omitted).
Query: Also "parallel measurement" port groups can be queried.

Parameters:
<lport0>, <lport1>, ... Logical port numbers. Must be omitted if the command is used as a query.

SOURce<Ch>:GROup<Grp>:PPORt<PhyPort>:DPORt <DestinationPort>

In "Set" direction this command connects physical VNA port <PhyPort> to DUT <Grp> port <DestinationPort>. The query returns the connected DUT port (if any).

Note

- The related port group must have been created before using SOURce<Ch>:GROup<Grp>:PPORts.
- It is not possible to connect VNA and DUT ports more than once.

Suffix:
<Ch> Channel number

<Grp> Port group number

<PhyPort> Number of a physical VNA port in port group <Grp>.

Parameters:

<DestinationPort> Number of a port on the related DUT.
Must be smaller than the size of the DUT <Grp> (set using SOURce<Ch>:GROup<Grp>:DPORt:COUNt).

Manual operation: See "Controls and Functions" on page 695

SOURce<Ch>:GROup<Grp>:PPORts <pport1>,<pport2>...

Defines port group <Grp> from physical VNA ports <pport1>, <pport2>, ...

Note

- Port groups must be created in ascending order, i.e. first create port group 1, then port group 2 etc.
- By default there is a single port group, which contains all physical VNA ports ("default port group").
- Redefinition of an existing port group is not allowed (exception: "default port group"). Use SOURce<Ch>:GROup<Grp>:CLEar to dissolve existing port groups first.
- It is not allowed to assign the same physical VNA port to multiple port groups.
- When a port group is created, the number of ports of the corresponding DUT is set to the port group size, and "aligned" port connections are established.
  The result can be modified using SOURce<Ch>:GROup<Grp>:DPORt:COUNt.

Suffix:

<Ch> Channel number

<Grp> Port group number

Parameters:

<pport1>,<pport2>... Physical VNA port numbers

Example:

```
SOURce1:GROup1:PPorts 1, 2
SOURce1:GROup2:PPorts 3, 4
SOURce1:GROup1:NAME 'DUT_1'
SOURce1:GROup2:NAME 'DUT_2'
```

Creates the parallel measurement setup implemented in the Define Parallel Measurement dialog.
Requires a 4-port instrument with Second internal generator.

Manual operation: See "Controls and Functions" on page 695

SOURce<Ch>:GROup<Grp>:SIMultaneous:FOFFset:CONDition

Queries the runtime state of the "parallel measurement with frequency offset" feature. The list below contains all possible return values of this query. The query waits until all previous commands are executed, because these commands can affect the operation state.

Returns an error, if channel <Ch> does not contain a trace. Without traces, no measurements are performed and hence no feature-specific information is available.

Suffix:

<Ch> Channel number

<Grp> Port group number

Return values:

<errorCode>, <errorText>

0, "switched off"
"Parallel measurement with frequency offset" is switched OFF, i.e. `SOURce<Ch>:GROup<Grp>:SIMultaneous:FOFFset:STATe?` is OFF.

1, "active"
"Parallel measurement with frequency offset" is active, meaning it has been activated via `SOURce<Ch>:GROup:SIMultaneous:FOFFset:STATe ON` and all preconditions for this mode to be operational are met.

-1, "too few port groups"
There must be at least two port groups defined, otherwise no parallel measurement is possible.

-2, "invalid port groups"
Set up the port groups as in the VNA GUI, when "Enabled" is selected in the Define Parallel Measurement dialog.

-4, "invalid sweep type"
"Parallel measurement with frequency offset" requires either a linear frequency sweep or a segmented sweep.

-6, "no parallel measurement possible"
The VNA did not manage to measure the port groups simultaneously.

-7, "stimulus buffer too small"
The stimulus buffer must contain at least two elements.

-8, "invalid offset or frequency spacing"
The frequency offset the firmware uses, is determined both by the minimum frequency offset and the frequency step size. If it is too large, the VNA is not able to measure simultaneously at different frequencies anymore.
There is no simple formula specifying the maximum frequency offset which can be handled by the VNA. However, in all but special cases an actual frequency offset of several MHz can be used. Note, however, that a small minimum frequency offset combined with a large frequency step size can lead to a large actual frequency offset.

-9, "extended stimulus buffer too large"
Internally, the firmware uses additional sweep points that are transparent to the user ("extended stimulus buffer"). The total number of sweep points (user-defined + additional) must not exceed the maximum number of sweep points (100001).
The "extended stimulus buffer" size depends on the relation between minimum frequency offset and frequency step size. The worst case occurs when the minimum frequency offset is equal to or greater than the sweep range. In this case, the extended stimulus buffer contains the number of sweep points entered by the user times the number of port groups.

Example: See `SOURce<Ch>:GROup<Grp>:SIMultaneous:FOFFset:STATe`

Manual operation: See "State indication warning" on page 698

SOURce<Ch>:GROup<Grp>:SIMultaneous:FOFFset:MOFFset:BWFactor <Value>

This command is related to "parallel measurement with frequency offset". It defines the multiplication factor that is used to calculate the minimum frequency offset from the measurement bandwidth.

This setting only takes effect if `SOURce<Ch>:GROup<Grp>:SIMultaneous:FOFFset:MOFFset:MODE` is set to `BANDwidth`.

Suffix:
<Ch> Channel number

<Grp> Port group number

Parameters:
<Value> The multiplication factor to calculate the minimum frequency offset.
To get the resulting frequency offset, the minimum frequency offset is rounded to a multiple of the current frequency step size.

Range: 1 to 100
*RST: 10

Example: See `SOURce<Ch>:GROup<Grp>:SIMultaneous:FOFFset:STATe`

Manual operation: See "Minimum Offset" on page 697

SOURce<Ch>:GROup<Grp>:SIMultaneous:FOFFset:MOFFset:DVALue
<MinFreqOffs>

This command is related to "parallel measurement with frequency offset". It specifies the minimum frequency offset directly.

This setting only takes effect if `SOURce<Ch>:GROup<Grp>:SIMultaneous:FOFFset:MOFFset:MODE` is set to `DIRect`.

Suffix:
<Ch> Channel number.

<Grp> Port group number

Parameters:
<MinFreqOffs> The minimum frequency offset.
To get the resulting frequency offset, the minimum frequency offset is rounded to a multiple of the current frequency step size.

Range: 0 Hz to 17 MHz
*RST: 1 MHz
Default unit: Hz

Example: See `SOURce<Ch>:GROup<Grp>:SIMultaneous:FOFFset:STATe`

Manual operation: See "Minimum Offset" on page 697

SOURce<Ch>:GROup<Grp>:SIMultaneous:FOFFset:MOFFset:MODE
<ModeEnum>

This command is related to "parallel measurement with frequency offset". It defines how the minimum frequency offset between different port groups is determined.

The firmware calculates the actual offset as the smallest multiple of the current frequency step size (sweep or segment) ≥ the minimum frequency offset. It can be larger than the specified minimum offset.

Suffix:
<Ch> Channel number.

<Grp> Port group number

Parameters:
<ModeEnum> DIRect | BANDwidth

DIRect
The minimum frequency offset to be used is directly specified by the command `SOURce<Ch>:GROup<Grp>:SIMultaneous:FOFFset:MOFFset:DVALue`.

BANDwidth
The minimum frequency offset to be used is calculated by the firmware as a product of the (possibly segment specific) measurement bandwidth and the factor specified by the command `SOURce<Ch>:GROup<Grp>:SIMultaneous:FOFFset:MOFFset:BWFactor`.
In this calculation of the minimum frequency offset, static and dynamic reduction of the measurement bandwidth is **not** considered. The firmware uses the measurement bandwidth as it has been specified via `[SENSe<Ch>:]BWIDth[:RESolution]` (for the entire sweep) or via `[SENSe<Ch>:]SEGMent<Seg>:BWIDth[:RESolution]` (segment specific).

*RST: BANDwidth

Example: See SOURce<Ch>:GROup<Grp>:SIMultaneous:FOFFset:STATe

Manual operation: See "Minimum Offset" on page 697

SOURce<Ch>:GROup<Grp>:SIMultaneous:FOFFset:STATe <arg0>

This command is used to activate/deactivate a frequency offset for parallel measurements on a R&S ZNB with Second internal generator.

The port groups must be defined using SOURce<Ch>:GROup<Grp>:PPORts.

Even if the parallel measurement is properly configured and this flag is set to ON, the frequency offset can still be inactive if not all preconditions are met. For each precondition, there is a unique return value for the query SOURce<Ch>:GROup<Grp>:SIMultaneous:FOFFset:CONDition which is returned when the according precondition is not met. A complete list of all preconditions can be found in the description of the command's return values. It is possible to first activate the frequency offset and afterwards issue other remote commands to set up all necessary preconditions.

Suffix:

<Ch> Channel number.
If unspecified, this parameter is set to 1.

<Grp>

Parameters:

<arg0> ON | OFF – activates or deactivates frequency shifted parallel measurements.

Example:

```
*RST
SENSe:FREQuency:STARt 1GHZ
SENSe:FREQuency:STOP 2GHZ
CALCulate:PARameter:MEASure 'Trc1', 'S21'
*WAI
```

Perform a reset and set up a trace (with the default of 201 sweep points).

```
:SOUR:GRO1:PPORTs 1,2
:SOUR:GRO2:PPORTs 3,4
:INIT:IMM;*WAI
```

Create two port groups with ports 1 and 2 in port group 1 and ports 3 and 4 in port group 2 and start the parallel measurement.

```
:SOUR:GRO1:NAME 'First DUT'
:SOUR:GRO2:NAME 'Second DUT'
```

Assign descriptive names to groups.

```
SOUR:GRO:COUN?
```

Check if two port groups exist.

```
:SOUR:GRO1:NAME?
:SOUR:GRO2:NAME?
```

Returns 'First DUT' and 'Second DUT', respectively.

```
:SOUR:GRO1:PPORT1:DPORt 1
:SOUR:GRO1:PPORT2:DPORt 2
:SOUR:GRO2:PPORT3:DPORt 2
:SOUR:GRO2:PPORT4:DPORt 1
:INIT:IMM;*WAI
```

Assign DUT Ports to VNA Ports.

```
CALCulate1:PARameter:SDEFine 'Trc2', 'S34'
DISPlay:WINDow1:TRACe2:FEED 'Trc2'
*WAI
```

Add and display a second trace for the second DUT (note that 'S34' with VNA port numbers corresponds to S21 at the second DUT.

```
SOURce:GROup:SIMultaneous:FOFFset:MOFFSet:
DVALue 6 MHz
SOURce:GROup:SIMultaneous:FOFFset:MOFFset:MODE
DIRect
```

Set minimum frequency offset for frequency shifted parallel measurement to 5 MHz.

```
SOURce:GROup:SIMultaneous:FOFFset:CONDition?
```

Returns `%0, "switched off"`

```
SOURce:GROup:SIMultaneous:FOFFset:STATe?
```

Returns `0`.

```
SOURce:GROup:SIMultaneous:FOFFset:STATe ON
```

Activate frequency shifted parallel measurement.

```
SOURce:GROup:SIMultaneous:FOFFset:CONDition
```

Check if frequency shifted parallel measurement is active (should return `1, "active"`).

```
*WAI
```

Wait until the previous commands are executed.

```
SOURce:GROup:SIMultaneous:FOFFset:MOFFSet:
DVALue 0 MHz
```
Set minimum frequency offset for frequency shifted parallel measurement to 0 MHz.
```
:SOURce:GROup:SIMultaneous:FOFFset:MOFFset:MODE
BANDwidth
:SOURce:GROup:SIMultaneous:FOFFset:MOFFset:
BWFactor 100
```
Set the minimum frequency offset for frequency shifted parallel measurement to at least 100 times the measurement bandwidth.

Manual operation: See "Parallel Measurement with Frequency Offset" on page 697

7.3.15.3 SOURce:NFIGure...

Calibration power and power leveling settings of Noise Figure (with Noise figure measurement option R&S ZNB-K30).

SOURce<Ch>:NFIGure:CALibration:APOWer <PowValue>

Defines the source power for measuring the external source attenuator (together with the Through standard) during calibration of noise figure channel <Ch>.

Suffix:
<Ch> Channel number

Parameters:
<PowValue> Default unit: dBm

Options: R&S ZNB-K30

Manual operation: See "External Attenuation" on page 355

SOURce<Ch>:NFIGure:CALibration:PPOWer <PowValue>

Defines the source power for measuring 1-port standards during calibration of noise figure channel <Ch>.

Suffix:
<Ch> Channel number

Parameters:
<PowValue> Default unit: dBm

Options: R&S ZNB-K30

Manual operation: See "Power (1-P Std)" on page 355

SOURce<Ch>:NFIGure:CALibration:TPOWer <PowValue>

Defines the source power for measuring Through standards during calibration of noise figure channel <Ch>.

Suffix:
<Ch> Channel number

Parameters:
<PowValue> Default unit: dBm

Options: R&S ZNB-K30

Manual operation: See "Power (Through)" on page 355

SOURce<Ch>:NFIGure:PLEVeling:DGAin <GainValue>

Defines the (maximum) gain of your DUT for channel <Ch>.

Suffix:
<Ch> Channel number

Parameters:
<GainValue> Default unit: dB

Options: R&S ZNB-K30

Manual operation: See "DUT Gain" on page 351

SOURce<Ch>:NFIGure:PLEVeling:DMAX <PowValue>

Defines the maximum source power of NF channel <Ch>.

Suffix:
<Ch> Channel number

Parameters:
<PowValue> Default unit: dBm

Options: R&S ZNB-K30

Manual operation: See "Driving Port Pmax" on page 351

SOURce<Ch>:NFIGure:PLEVeling:FATTenuation <AttValue>

For NF channel <Ch>, this setting defines the fixed attenuation at DUT input.

Suffix:
<Ch> Channel number

Parameters:
<AttValue> Default unit: dB

Options: R&S ZNB-K30

Manual operation: See "Fixed Attenuation" on page 351

SOURce<Ch>:NFIGure:PLEVeling:LGAin <GainValue>

Defines the gain of the LNA at the receiving VNA port for channel <Ch>.

Suffix:
<Ch> Channel number

Parameters:
<GainValue> Default unit: dBm

Options: R&S ZNB-K30

Manual operation: See "LNA Gain" on page 351

SOURce<Ch>:NFIGure:PLEVeling:RATTenuation <AttValue>

Defines the fixed attenuation at DUT output for NF channel <Ch>.

Suffix:
<Ch> Channel number

Parameters:
<AttValue> Default unit: dB

Options: R&S ZNB-K30

Manual operation: See "Receiving Attenuation" on page 351

SOURce<Ch>:NFIGure:PLEVeling:RMAX <PowValue>

Defines the intended maximum power at the receiving VNA port for NF channel <Ch>.

Suffix:
<Ch> Channel number

Parameters:
<PowValue> Default unit: dBm

Options: R&S ZNB-K30

Manual operation: See "Reveiving Port Pmax" on page 351

SOURce<Ch>:NFIGure:PLEVeling:SATTenuation <AttValue>

Defines the source attenuation of NF channel <Ch>.

Suffix:
<Ch> Channel number

Parameters:
<AttValue> Default unit: dB

Options: R&S ZNB-K30

Manual operation: See "Source Attenuation" on page 351

7.3.15.4 SOURce:POWer:CORRection:COLLect...

The `SOURce:POWer:CORRection:COLLect...` commands allow you to set up a scalar source power calibration.

SOURce:POWer:CORRection:COLLect:AVERage[:COUNt] <NoReadings>

Sets a limit for the number of calibration sweeps in the source power calibration. The command is valid for all channels, ports and external generators.

Equivalent command: `SOURce:POWer:CORRection:NREadings`.

Parameters:

<NoReadings> Number of readings

Range: 1 to 100
*RST: 2

Example: See `SOURce<Ch>:POWer:CORRection[:ACQuire]` (for equivalent command `SOURce:POWer:CORRection:NREadings`)

Manual operation: See "Flatness Cal – Max Iterations" on page 649

SOURce:POWer:CORRection:COLLect:AVERage:NTOLerance <Tolerance>

Specifies the maximum deviation of the measured power from the target power of the calibration. The command is valid for all channels and calibrated ports.

Parameters:

<Tolerance> Tolerance value

Range: 0.001 dB to 1000 dB
*RST: 1 dB
Default unit: dB

Example: See `SOURce<Ch>:POWer:CORRection[:ACQuire]`

Manual operation: See "Flatness Cal – Tolerance" on page 649

SOURce:POWer:CORRection:COLLect:CFACtor <Convergence>

Specifies the convergence factor for a source power calibration.

Parameters:

<Convergence> Convergence factor

Range: 0 to 2
Increment: 0.1
*RST: 0.8

Example: See `SOURce<Ch>:POWer:CORRection[:ACQuire]`

Manual operation: See "Flatness Cal – Convergence" on page 649

SOURce:POWer:CORRection:COLLect:FLATness <Boolean>

Enables or disables the source power calibration. The command is valid for all channels, ports and external generators.

It is not possible to disable flatness calibration and the reference receiver calibration simultaneously (see SOURce:POWer:CORRection:COLLect:RRECeiver on page 1502).

Parameters:

<arg0> <Boolean>

Disables (OFF|0) or enables (ON|1) the flatness calibration. With disabled flatness calibration, only one calibration sweep is performed in order to calibrate the reference receiver; the previous source calibration data is not overwritten.

*RST: ON|1

Example:

```
*RST; SOUR:POW:CORR:COLL:VER 0
```

Disable the verification sweep to speed up the source power calibration procedure.

```
SOUR:POW:CORR:COLL:FLAT OFF; RREC ON
```

Disable flatness cal and enable reference receiver calibration.

```
SOUR:POW:CORR:ACQ PORT,1
```

Perform a single source power calibration sweep using port 1 as a source port.

SOURce:POWer:CORRection:COLLect:METHod <PowerCalMethod>

Selects the source power calibration method.

Parameters:

<PowerCalMethod> PMONly | RRAFter | RRONly

PMONly
Power meter only, i.e. only the power meter is used during calibration.
To enable a reference reveiver and/or source flatness calibration, set SOURce:POWer:CORRection:COLLect:RRECeiver and/or SOURce:POWer:CORRection:COLLect:FLATness to ON.

RRAFter
First the reference receiver will be calibrated by a single power meter measurement, then the source flatness calibration will be performed using the calibrated reference receiver.
Requires SOURce<Ch>:POWer<PhyPt>:CORRection:COLLect:FLATness ON.

RRONly
Reference receiver only. The reference receiver will be used to perform a source flatness calibration. Ideally the reference receiver was previously calibrated using a power meter.
Requires `SOURce<Ch>:POWer<PhyPt>:CORRection:COLLect:RRECeiver OFF`.

*RST: RRAFter

Example: See `SOURce<Ch>:POWer:CORRection[:ACQuire]`

Manual operation: See "Flatness Cal – Pwr Cal Method" on page 649

SOURce:POWer:CORRection:COLLect:RRECeiver <Boolean>

Enables or disables calibration of the reference receiver together with the source power calibration.

Note: It is not possible to disable flatness calibration (see `SOURce:POWer:CORRection:COLLect:FLATness` on page 1501) and reference receiver calibration simultaneously.

Parameters:

<Boolean> Disables (OFF|1) or enables (ON|1) reference receiver calibration.

*RST: ON|1

Example: See `SOURce:POWer:CORRection:COLLect:FLATness` on page 1501.

SOURce:POWer:CORRection:COLLect:VNETworks <DeEmbNwType>[, <DeEmbType>]

Defines whether the source flatness cal uses the configured offset de-/embedding to get the power values at the DUT (instead of the calibrated reference plane).

This setting applies to all physical ports and all channels.

Parameters:

<DeEmbNwType> SENDed | GLOop | OFF | ALL | DIFFerential | PSET | RENorm | OFFSet | AOFFset

Selects the offset or de-/embedding type to be used, if configured.
ALL means that all available offset and de-/embedding information is applied. The <DeEmbType> can be omitted in this case.

*RST: OFF

<DeEmbType> EMBedding | DEEMbedding

Selects between embedding and deembedding for the selected <DeEmbNwType>.
Must be specified for network types that can be used for embedding and deembedding

Manual operation: See "Embedding" on page 650

7.3.15.5 SOURce:POWer:CORRection:TCOefficient...

These commands allow you to modify the results of a scalar power calibration to account for additional two-port devices (with known transmission characteristics) in the test setup.

SOURce<Ch>:POWer<PhyPt>:CORRection:TCOefficient:CALibration <Boolean>

Selects the position of the additional two-port in the test setup.

Note that it is now possible to have two-ports at both positions (see `SOURce:POWer:CORRection:TCOefficient[:STATe]`). This command is kept for compatibility reasons.

Suffix:

<Ch> Calibrated channel number. This parameter is ignored; the transmission coefficient settings are valid for all channels.

<PhyPt> Calibrated port number. This parameter is ignored; the transmission coefficient settings are valid for all sources.

Parameters:

<Boolean> ON – two port at power meter (during calibration)
OFF – two port at DUT (during measurement)

*RST: ON (notice that the correction is switched off after a reset)

Example:
```
RST; :SOUR:POW:CORR:TCO:CAL OFF
```
Select the test setup with the additional two-port in front of the DUT.
```
SOUR:POW:CORR:TCO:DEF 1GHz, -5; DEF 2GHz, -10; DEF2?
```
Define two points in the power loss list; query the second point.
```
SOUR:POW:CORR:TCO:INS2 1.5 GHz, -7.5
```
Insert a third point as point no. 2 in the list.

Example:
```
SOUR:POW:CORR:TCO:COUN?
```
Query the number of points. The response is `3`.
```
SOUR:POW:CORR:TCO ON
```
Enable the use of two-port transmission coefficients.
```
SOUR:POW:CORR:TCO:FEED 'Trc1'
```
Replace the previous 3 points by the trace points of the default trace "Trc1".

Example: `MMEM:STOR:CORR:TCO 'C:\Users\Public\Documents\Rohde-Schwarz\Vna\PowerMeterCorr\Test.pmcl'`
Store the power loss list to the power meter correction list file `Test.pmcl` in the directory `C:\Users\Public\Documents\Rohde-Schwarz\Vna\PowerMeterCorr`.
`MMEM:LOAD:CORR:TCO 'C:\Users\Public\Documents\Rohde-Schwarz\Vna\PowerMeterCorr\Test.pmcl'`
Reload the stored power meter correction list file.

Example: `SOUR:POW:CORR:TCO:DEL1`
Delete the first point in the list.
`SOUR:POW:CORR:TCO:DEL:ALL; :SOUR:POW:CORR:TCO?`
Clear the entire list. Query whether the transmission coefficients are still used. The response is `0`.
`MMEM:LOAD:CORR:TCO 'C:\Users\Public\Documents\Rohde-Schwarz\Vna\Traces\Test.s2p', 'S21'`
Load a power loss list from a previously created 2-port Touchstone file.

SOURce:POWer:CORRection:TCOefficient:COUNt? [<DeEmbeddingPosition>]

Queries the number of entries in the transmission coefficient list.

Query parameters:

<DeEmbeddingPosition> BOTH | DUT | PM

Determines the two-port the command refers to:

DUT
The two-port between VNA and DUT (during measurement)

PM
The two-port between VNA and PM (during power calibration)

BOTH
Both positions

(parameter omitted)
If the parameter is omitted, the command refers to the two-port selected using `SOURce<Ch>:POWer<PhyPt>:CORRection:TCOefficient:CALibration`.

Example: See `SOURce:POWer:CORRection:TCOefficient[:STATe]` on page 1508

Usage: Query only

Manual operation: See "Frequency / Transm. Coefficients: Insert, Delete, Delete All" on page 654

SOURce:POWer:CORRection:TCOefficient:DEFine<ListNo> <Frequency>, <TransCoeff>[, <DeEmbeddingPosition>]

Adds a new frequency and transmission coefficient to the end of the transmission coefficient list. The query returns the frequency and transmission coefficient no. `<ListNo>`.

Suffix:
<ListNo> List entry no. (only used for queries)

Parameters:
<Frequency> Stimulus frequency value. If several points with identical frequencies are added, the analyzer automatically ensures a frequency spacing of 1 Hz.

Range: Stimulus values outside the frequency range of the analyzer are allowed.
*RST: n/a
Default unit: Hz

<TransCoeff> Transmission coefficient

Range: -300 dB to +200 dB
*RST: n/a
Default unit: dB

<DeEmbeddingPosition> BOTH | DUT | PM

Determines the two-port the command refers to:

DUT
The two-port between VNA and DUT (during measurement)

PM
The two-port between VNA and PM (during power calibration)

BOTH
Both positions

(parameter omitted)
If the parameter is omitted, the command refers to the two-port selected using `SOURce<Ch>:POWer<PhyPt>:CORRection:TCOefficient:CALibration`.

Example: See `SOURce:POWer:CORRection:TCOefficient[:STATe]` on page 1508

Manual operation: See "Frequency / Transm. Coefficients: Insert, Delete, Delete All" on page 654

SOURce:POWer:CORRection:TCOefficient:DELete:ALL [<DeEmbeddingPosition>]

Clears the transmission coefficient list.

Setting parameters:
<DeEmbeddingPosition> BOTH | DUT | PM

Determines the two-port the command refers to:

DUT
The two-port between VNA and DUT (during measurement)

PM
The two-port between VNA and PM (during power calibration)

BOTH
Both positions

(parameter omitted)
If the parameter is omitted, the command refers to the two-port selected using `SOURce<Ch>:POWer<PhyPt>:CORRection:TCOefficient:CALibration`.

Example: See `SOURce:POWer:CORRection:TCOefficient[:STATe]` on page 1508

Usage: Setting only

Manual operation: See "Frequency / Transm. Coefficients: Insert, Delete, Delete All" on page 654

SOURce:POWer:CORRection:TCOefficient:DELete<ListNo>[:DUMMy]
[<DeEmbeddingPosition>]

Deletes entry `<ListNo>` from the transmission coefficient list.

Suffix:

<Ch> Calibrated channel number. This suffix is ignored; the transmission coefficient settings are valid for all channels.

<PhyPt> Calibrated port number. This suffix is ignored; the transmission coefficient settings are valid for all sources.

<ListNo> List entry no.

Parameters:

<DeEmbeddingPosition> BOTH | DUT | PM

Determines the two-port the command refers to:

DUT
The two-port between VNA and DUT (during measurement)

PM
The two-port between VNA and PM (during power calibration)

BOTH
Both positions

(parameter omitted)
If the parameter is omitted, the command refers to the two-port selected using `SOURce<Ch>:POWer<PhyPt>:CORRection:TCOefficient:CALibration`.

Example: See `SOURce:POWer:CORRection:TCOefficient[:STATe]` on page 1508

Manual operation: See "Frequency / Transm. Coefficients: Insert, Delete, Delete All" on page 654

SOURce:POWer:CORRection:TCOefficient:FEED <Trace>[, <DeEmbeddingPosition>]

Selects a trace which provides the points for the transmission coefficient list.

Setting parameters:

<Trace> String parameter for the trace name, e.g. 'Trc1'. The trace must exist in the active recall set.

*RST: n/a

<DeEmbeddingPosition> BOTH | DUT | PM

Determines the two-port the command refers to:

DUT
The two-port between VNA and DUT (during measurement)

PM
The two-port between VNA and PM (during power calibration)

BOTH
Both positions

(parameter omitted)
If the parameter is omitted, the command refers to the two-port selected using `SOURce<Ch>:POWer<PhyPt>:CORRection:TCOefficient:CALibration`.

Example: See `SOURce:POWer:CORRection:TCOefficient[:STATe]` on page 1508

Usage: Setting only

Manual operation: See "Get Trace" on page 654

SOURce:POWer:CORRection:TCOefficient:INSert<ListNo> <Frequency>, <TransCoeff>[, <DeEmbeddingPosition>]

Inserts a new frequency and transmission coefficient at position no. <ListNo> in the transmission coefficient list. The following points are shifted down in the list.

Suffix:

<ListNo> List entry no.

Parameters:

<Frequency> Stimulus frequency value. If several points with identical frequencies are added, the analyzer automatically ensures a frequency spacing of 1 Hz.

Range: Stimulus values outside the frequency range of the analyzer are allowed.
*RST: n/a
Default unit: Hz

<TransCoeff> Transmission coefficient

Range: -300 dB to +200 dB
*RST: n/a
Default unit: dB

<DeEmbeddingPosition> BOTH | DUT | PM

Determines the two-port the command refers to:

DUT
The two-port between VNA and DUT (during measurement)

PM
The two-port between VNA and PM (during power calibration)

BOTH
Both positions

(parameter omitted)
If the parameter is omitted, the command refers to the two-port selected using `SOURce<Ch>:POWer<PhyPt>:CORRection:TCOefficient:CALibration`.

Example: See `SOURce:POWer:CORRection:TCOefficient[:STATe]` on page 1508

Manual operation: See "Frequency / Transm. Coefficients: Insert, Delete, Delete All" on page 654

SOURce:POWer:CORRection:TCOefficient[:STATe] <Boolean>[, <DeEmbeddingPosition>]

Enables or disables the use of two-port transmission coefficients for power correction.

Parameters:

<Boolean> Enables (ON) or disables (OFF) the two-port transmission coefficients.

*RST: n/a

<DeEmbeddingPosition> BOTH | DUT | PM

Determines the two-port the command refers to:

DUT
The two-port between VNA and DUT (during measurement)

PM
The two-port between VNA and PM (during power calibration)

BOTH
Both positions

(parameter omitted)
If the parameter is omitted, the command refers to the two-port selected using `SOURce<Ch>:POWer<PhyPt>:CORRection:TCOefficient:CALibration`.

Example:

```
RST; SOUR:POW:CORR:TCO:DEF 1GHz, -5, DUT; DEF 2GHz, -10, DUT; DEF2?
```
Define two points in the power loss list of the two-port in front of the DUT; query the second point.
```
SOUR:POW:CORR:TCO:INS2 1.5 GHz, -7.5, DUT
```
Insert a third point as point no. 2 in the list.
```
SOUR:POW:CORR:TCO:COUN? DUT
```
Query the number of points. The response is `3`.
```
SOUR:POW:CORR:TCO ON, DUT
```
Enable the use of two-port transmission coefficients.
```
SOUR:POW:CORR:TCO:FEED 'Trc1', DUT
```
Replace the previous 3 points by the trace points of the default trace "Trc1".
```
MMEM:STOR:CORR:TCO 'C:\Users\Public\Documents\Rohde-Schwarz\Vna\PowerMeterCorr\Test.pmcl', DUT
```
Store the power loss list to the power meter correction list file `Test.pmcl` in the directory `C:\Users\Public\Documents\Rohde-Schwarz\Vna\PowerMeterCorr`.
```
MMEM:LOAD:CORR:TCO 'C:\Users\Public\Documents\Rohde-Schwarz\Vna\PowerMeterCorr\Test.pmcl', DUT
```
Re-load the stored power meter correction list file.
```
SOUR:POW:CORR:TCO:DEL1, DUT
```
Delete the first point in the list.
```
SOUR:POW:CORR:TCO:DEL:ALL, DUT; :SOUR:POW:CORR:TCO? DUT
```
Clear the entire list. Query whether the transmission coefficients are still taken into account. The response is `0`.
```
MMEM:LOAD:CORR:TCO 'C:\Users\Public\Documents\Rohde-Schwarz\Vna\Traces\Test.s2p', 'S21', DUT
```
Load a power loss list from a previously created 2-port Touchstone file.

Manual operation: See "Active" on page 653

SOURce<Ch>:POWer<PhyPt>:CORRection:TCOefficient:SWAP <Boolean>, <DeEmbeddingPosition>

Flips the gate assignments if the network is specified by a two-port S-parameter file.

Suffix:

<Ch> Calibrated channel number. This suffix is ignored; the transmission coefficient settings are valid for all channels.

<PhyPt> Calibrated port number. This suffix is ignored.

Parameters:

<Boolean> Enables (ON) or disables (OFF) the two-port transmission coefficients.

<DeEmbeddingPosition> BOTH | DUT | PM

Determines the two-port the command refers to:

DUT
The two-port between VNA and DUT (during measurement)

PM
The two-port between VNA and PM (during power calibration)

BOTH
Both positions

7.3.15.6 SOURce:POWer:CORRection... (other)

The `SOURce:POWer:CORRection...` commands control a scalar source power calibration.

SOURce<Ch>:POWer:CORRection[:ACQuire] 1510
SOURce<Ch>:POWer<PhyPt>:CORRection[:ACQuire]:VERification:RESult? 1512
SOURce<Ch>:POWer:CORRection:DATA 1513
SOURce<Ch>:POWer:CORRection:DATA:PORT<PhyPt> 1513
SOURce<Ch>:POWer:CORRection:DATA:PARameter<Wv>? 1513
SOURce<Ch>:POWer<PhyPt>:CORRection:DATA:PARameter<Wv>:COUNt? 1515
SOURce<Ch>:POWer<PhyPt>:CORRection:GENerator<Gen>:LEVel:OFFSet 1515
SOURce<Ch>:POWer:CORRection:GENerator<Gen>[:STATe] 1516
SOURce<Ch>:POWer<PhyPt>:CORRection:IMODulation:LTONe[:ACQuire] 1517
SOURce<Ch>:POWer:CORRection:IMODulation:PORT 1517
SOURce<Ch>:POWer<PhyPt>:CORRection:IMODulation:UTONe[:ACQuire] 1518
SOURce<Ch>:POWer<PhyPt>:CORRection:LEVel:OFFSet 1518
SOURce<Ch>:POWer:CORRection:NISTandards:DEFine 1519
SOURce<Ch>:POWer:CORRection:NISTandards:DELete 1520
SOURce<Ch>:POWer:CORRection:NISTandards:LOAD 1520
SOURce:POWer:CORRection:NREadings 1521
SOURce<Ch>:POWer:CORRection:OSOurces[:STATe] 1521
SOURce:POWer:CORRection:PMETer:ID 1521
SOURce:POWer:CORRection:PPOWer 1522
SOURce:POWer:CORRection:PSELect 1522
SOURce<Ch>:POWer<PhyPt>:CORRection:STATe 1523

SOURce<Ch>:POWer:CORRection[:ACQuire] <SourceType>[, <SourcePort>[, <CalOnlyPortFreq>]]

Selects the source for the source power calibration, starts and applies the source power calibration.

Note: The command cannot be used unless a power meter is connected via GPIB bus, USB or LAN interface and configured in the "External Power Meters" dialog.

Suffix:

<Ch> Calibrated channel number

Setting parameters:

<SourceType> PORT | GENerator | A1 | A2 | A3 | A4 | ESRC1 | ESRC2

PORT – analyzer port is the source of the calibrated wave.
GENerator – external generator is the source.
The port and generator numbers are specified by means of the `<SourcePort>` parameter.
A1 | A2 | A3 | A4 – direct analyzer port selection. The parameters A1 and A2 are ZVR-compatible. No additional port number parameter needs to be specified.
ESRC1 | ESRC2 – direct external generator selection. The parameters are ZVR-compatible. No additional parameters need to be specified.

<SourcePort> Analyzer port or generator port number, if PORT or GENerator is the first parameter.

<CalOnlyPortFreq> Relevant for frequency conversion measurements only.

OFF | 0
The reference receiver at port <CalPort> is calibrated for all frequencies that are relevant for any of the ports.

ON | 1
Only those frequencies are calibrated that are relevant for this port. This results in shorter calibration times.

*RST: 0

Example:

```
SOUR:POW:CORR:OSO:STAT OFF
```
To improve the accuracy, switch off all other sources during the calibration sweep.
```
SOUR:POW:CORR:NRE 2; COLL:AVER:NTOL 0.5
```
Increase the number of readings and reduce the power tolerance to improve the accuracy.
```
SOURce:POW:CORR:COLL:CFAC 0.9
```
Reduce the correction factor to 0.9.
```
SOURce:POW:CORR:PSELect PPOW; PPOWer -5
```
Define reference receiver cal power which is independent of the port power; set its value to -5 dBm.
```
SOUR:POW:CORR:ACQ PORT,1
```
Perform a source power calibration using port 1 as a source port.
```
SOUR:POW:CORR:STAT?
```
Check whether the calibration is applied (the response is `1`).

Example: Change the test setup, connect a previously configured external generator no. 1 to the reference plane.
`SOUR:POW:CORR:ACQ GEN,1`
Perform a source power calibration using the external generator no. 1 as a source.
`SOUR:POW:CORR:GEN?`
Check whether the calibration is applied (the response is `1`).

Usage: Setting only

Manual operation: See "Start Cal Sweep" on page 342

SOURce<Ch>:POWer<PhyPt>:CORRection[:ACQuire]:VERification:RESult?

Returns (and deletes) the result of the last verification sweep. The response contains three values:

- <Channel> – calibrated channel, e.g. `Ch 1` for channel no. 1.
- <Boolean> – 1 for "calibration passed" (the maximumr offset is below the tolerance), 0 for "calibration failed".
- <MaxOffset> – maximum power offset between the measured power at the reference plane and the "Cal Power" during the verification sweep in dB.

If no verfication sweep is available, or if the result has been queried already, the response is `0`.

Suffix:

<Ch> Calibrated channel number. This suffix is ignored; the result is based on the last verification sweep acquired.

<PhyPt> Calibrated port number. This suffix is ignored; the result is based on the last verification sweep acquired.

Example:
```
*RST; :SOURce:POWer:CORRection:NREadings 2
SOURce:POWer:CORRection:COLLect:AVERage:
NTOLerance 0.5
```
Increase the number of readings and reduce the power tolerance to improve the accuracy.
`SOURce:POWer:CORRection:ACQuire PORT,1`
Perform a source power calibration using port 1 as a source port.
`SOURce:POWer:CORRection:ACQuire:VERification:RESult?`
Query the result of the verification sweep. Possible response:
`Ch1 1,9.9E+002`

Usage: Query only

Manual operation: See "Calibration Sweep Diagram" on page 596

SOURce<Ch>:POWer:CORRection:DATA <Source>, <CorrData>...
SOURce<Ch>:POWer:CORRection:DATA:PORT<PhyPt> <Source>, <CorrData>...

Reads or writes scalar source power correction data sets. A power correction data set contains n real values where:

- Each value corresponds to the ratio of the actual power at the reference plane (value provided by the used source) to the uncalibrated power in dB.
- The number n is equal to the number of sweep points.

Increasing (decreasing) the values in the correction data sets increases (decreases) the power at the reference plane. Writing correction data (the setting command) fails if the number of transferred values is not equal to the number of sweep points.

Suffix:

<Ch> Calibrated channel number

<PhyPt> Physical port number, used to select a specific frequency axis in arbitrary mode (with option R&S ZNB-K4).

Parameters:

<Source> Identifier for the source of the calibrated wave:
'A<n>' or 'PORT<n>' denote correction data for the analyzer port no. <n>.
'ESRC<n>' or 'GEN<n>' denote correction data for the external generator no. <n>.

<CorrData> Power correction values either in ASCII or block data format, depending on the current FORMat[:DATA] setting.
All numbers are interpreted as dB-values.

Example:

```
*RST; :SWE:POIN 10
```
Reset the instrument and reduce the number of sweep points to 10.
```
SOUR:POW:CORR:ACQ PORT,1
```
Perform a source power calibration using port 1 as a source port.
```
SOUR:POW:CORR:DATA? 'A1'
```
Query the correction values. The analyzer returns 10 comma-separated real numbers.
```
SOUR:POW:CORR:DATA 'A1', 1, 2, 3, 4, 5, -6, -7, -8, -9, -0
```
Replace the correction values by 10 (new) numbers.

Manual operation: See "Apply" on page 598

SOURce<Ch>:POWer:CORRection:DATA:PARameter<Wv>?
<PowerCalSettingType>[, <Index>]

Gets settings of the available power calibrations.

Suffix:

<Ch> Number of the calibrated channel.

<Wv> Number of the power calibration.
Use SOURce<Ch>:POWer<PhyPt>:CORRection:DATA:PARameter<Wv>:COUNt? on page 1515 to query the number of available power calibrations.

Query parameters:

SettingType WAVE | STARt | STOP | POINts | STYPe | ATTenuation | CPOWer | CFRequency | TSTamp | LTSTamp | TVNA | MVNA | MTESt | FSMode

WAVE
Calibrated port and calibration type (Gen=Source Power, Rec=Measurement Receiver)

STARt
Start Frequency or Power

Stop Frequency or Power

POINts
Nr. of sweep points

STYPe
Sweep type or grid (LIN, LOG, SEGM)

ATTenuation
Source or receiver attenuation

CPOWer
Cal Power for frequency sweeps

CFRequency
Cal Frequency for power sweeps

TSTamp
Timestamp in UTC

LTSTamp
Timestamp in local time

TVNA
For calibrations involving switch matrices, this indicates the test ports on the VNA itself. Returns a comma-separated list of port pairs `TestPort,VnaPort`

MVNA
For calibrations involving switch matrices, this indicates the mapping between matrix ports and VNA ports for matrix <Index>. Returns a comma-separated list of port pairs `MatrixVnaPort,VnaPort`

MTESt
For calibrations involving switch matrices, this indicates the mapping between matrix (physical) test ports and test ports for matrix <Index>. Returns a comma-separated list of port pairs `MatrixTestPort,TestPort`

FSMode
Returns the frequency sweep mode that was used during calibration.
STEP: Stepped mode (for all segments)
ANAL: Swept mode (for at least one segment)
Because swept mode is limited to frequency sweeps this always returns STEP.

<Index> If one or more external switch matrices were used during calibration, this refers to the index of a particular switch matrix (see `SYSTem:COMMunicate:RDEVice:SMATrix<Matr>:DEFine` on page 1559)

Usage: Query only

Manual operation: See "Port Overview" on page 597

SOURce<Ch>:POWer<PhyPt>:CORRection:DATA:PARameter<Wv>:COUNt?

Gets the number of available power calibrations (both source power and measurement receiver).

Use `SOURce<Ch>:POWer:CORRection:DATA:PARameter<Wv>?` to retrieve the related settings.

Suffix:
<Ch> Number of the calibrated channel.

<PhyPt> This suffix is ignored.

<Wv> This suffix is ignored.

Usage: Query only

Manual operation: See "Port Overview" on page 597

SOURce<Ch>:POWer<PhyPt>:CORRection:GENerator<Gen>:LEVel:OFFSet
<Offset>

Specifies a gain (positive values) or an attenuation (negative values) in the signal path between the external generator and the calibrated reference plane. The value has no impact on the generator power.

Suffix:
<Ch> Calibrated channel number

<PhyPt> Calibrated port number. This suffix is ignored; the generator is selected via `<Gen>`.

<Gen> Generator number

Parameters:

<Offset> Gain or attenuation value

Range: -300 dB to +300 dB (adjust to the test setup)
Increment: 0.01 dB (other values are rounded)
*RST: 0 dB
Default unit: dB

Example: Assume that a DUT requires a constant input power of +35 dBm at its port 2, and that the measurement path contains an amplifier with a 30 dB gain.

```
SYST:COMM:RDEV:GEN1:DEF 'Ext. Gen. 1', 'SME02', 'gpib0', '21'
```

Configure an R&S SME02 generator as external generator no. 1, assigning the name Ext. Gen. 1 and a GPIB address 21.

```
SOUR:POW:GEN1:OFFS 5, ONLY
```

Set the source power of generator no. 1 to +5 dBm.

```
SOUR:POW:CORR:GEN1:LEV:OFFS 30
```

Specify the gain of the amplifier in the signal path between the external generator and the input port of the DUT.

```
SOUR:POW:CORR:ACQ GEN,1
```

Perform a source power calibration using the external generator no. 1 as a source port and the target power of +35 dBm.

Manual operation: See "Cal Power Offset" on page 652

SOURce<Ch>:POWer:CORRection:GENerator<Gen>[:STATe] <Boolean>

Enables or disables the source power calibration for channel <Ch> and for an external generator number <Gen>. The command is disabled unless a source power calibration for the external generator has been performed (SOURce<Ch>:POWer:CORRection[:ACQuire]). To enable or disable a source power calibration for an analyzer port use SOURce<Ch>:POWer<PhyPt>:CORRection:STATe.

Suffix:

<Ch> Calibrated channel number

<Gen> Generator number

Parameters:

<Boolean> Enables (ON) or disables (OFF) the source power calibration for port number <PhyPt>.

*RST: OFF

Example: See SOURce<Ch>:POWer:CORRection[:ACQuire]

Manual operation: See "Port Overview" on page 657

SOURce<Ch>:POWer<PhyPt>:CORRection:IMODulation:LTONe[:ACQuire]

Starts the source calibration for the lower tone (1st power calibration step for intermodulation measurements), stores and applies the calibration data. The external power meter used is selected via `SOURce<Ch>:POWer<PhyPt>:CORRection:PMETer:ID`.

Suffix:

<Ch> Calibrated channel number

<PhyPt> Port number This suffix is ignored; the analyzer calibrates the port that is selected as a source port for the intermodulation measurement (`[SENSe<Ch>:]FREQuency:IMODulation:LTONe`).

Example:

```
*RST; FREQ:IMOD:LTON PORT,1; UTON GEN, 1
```
Reset the instrument to create the default channel no. 1. Select port 1 as a source port for the lower tone, the (previously configured) external generator no. 1 as a source port for the upper tone.
```
FREQ:IMOD:REC 2
```
Select port 2 as a receiver port.
```
FREQ:STAR 1GHZ; STOP 2GHz
```
Define a suitable sweep range to ensure that the analyzer can measure all intermodulation products.
```
SOUR:POW:CORR:PMET:ID 1
```
Select power meter no. 1 (previously configured in the "External Power Meters" dialog and properly connected) for the RF source calibration.
```
SOUR:POW:CORR:IMOD:LTON
```
Start source power calibration for the lower tone at the previously selected source port 1.
```
SOUR:POW:CORR:IMOD:UTON
```
Start source power calibration for the upper tone at the previously selected source port 3.
```
CORR:POW:IMOD:ACQ
```
Start receiver power calibration at the previously selected receive port 2.
```
CALC:PAR:SDEF 'IM_Meas', 'IP3UI'
```
Create a new channel no. 2 and a trace named 'IM_Meas'. Select the upper 3rd-order intercept point at the DUT input as a measured quantity.

Usage: Event

Manual operation: See "Start Cal Sweep" on page 342

SOURce<Ch>:POWer:CORRection:IMODulation:PORT <SourceForRecCal>

Selects the source port for the receiver power calibration of an intermodulation measurement. An external generator must be configured before the generator port is available (`SYSTem:COMMunicate:RDEVice:GENerator<Gen>:DEFine`).

Suffix:
<Ch> Channel number

Parameters:
<SourceForRecCal> String parameter to identify the source port:
'COMBINER' – combiner output
'PORT <n>' – analyzer port n (separate port no. by blank, e.g. 'PORT 4')
'GEN <n>' – external generator port n (separate port no. by blank, e.g. 'GEN 1')

*RST: 'COMBINER'

Example: `*RST; :SOURce:POWer:CORRection:IMODulation:PORT 'PORT 4'`
Select the analyzer port no. 4 as a source port for the receiver power calibration.

Manual operation: See "Src. for Rcvr. Cal" on page 339

SOURce<Ch>:POWer<PhyPt>:CORRection:IMODulation:UTONe[:ACQuire] [<CalOnlyPortFreq>]

Starts the source calibration for the upper tone (2nd power calibration step for intermodulation measurements), stores and applies the calibration data. The external power meter used is selected via `SOURce<Ch>:POWer<PhyPt>:CORRection:PMETer:ID`.

Suffix:
<Ch> Calibrated channel number

Setting parameters:
<CalOnlyPortFreq>

OFF | 0
The reference receiver at port <CalPort> is calibrated for all frequencies that are relevant for any of the ports.

ON | 1
Only those frequencies are calibrated that are relevant for this port. This results in shorter calibration times.

*RST: 0

Example: See `SOURce<Ch>:POWer<PhyPt>:CORRection:IMODulation:LTONe[:ACQuire]`

Usage: Setting only

Manual operation: See "Start Cal Sweep" on page 342

SOURce<Ch>:POWer<PhyPt>:CORRection:LEVel:OFFSet <Offset>

Specifies a gain (positive values) or an attenuation (negative values) in the signal path between the source port and the calibrated reference plane. The value has no impact on the source power.

Suffix:
<Ch> Calibrated channel number

<PhyPt> Calibrated port number

Parameters:
<Offset> Gain or attenuation value

Range: -300 dB to +300 dB (adjust to the test setup)
Increment: 0.01 dB (other values are rounded)
*RST: 0 dB
Default unit: dB

Example: Assume that a DUT requires a constant input power of +35 dBm at port 2, and that the measurement path contains an amplifier with a 30 dB gain.

```
*RST; SOUR:POW 0
```

Reset the instrument and set the internal source power (base channel power) for the default channel 1 to 0 dBm.

```
SOUR:POW2:OFFS 5, ONLY
```

Replace the source power at port 2 by +5 dBm. The powers at the remaining ports are not affected.

```
SOUR:POW2:CORR:LEV:OFFS 30
```

Specify the gain of the amplifier in the signal path between the analyzer port 2 and the input port of the DUT.

```
SOUR:POW:CORR:ACQ PORT,2
```

Perform a source power calibration using port 2 as a source port and the target power of +35 dBm.

Manual operation: See "Cal Power Offset" on page 652

SOURce<Ch>:POWer:CORRection:NISTandards:DEFine <CorrData>...

Defines a comma-separated list of <frequency [Hz], gain [dB]> pairs. This list is written into an internal buffer and then applied for correcting non-ideal standards during subsequent measurement receiver power calibrations in channel <Ch>. See "Calibration with non-ideal standards" on page 190.

The frequencies must "cover" the calibrated frequency range. If they do not coincide with the calibration sweep points, linear interpolation is used.

Suffix:
<Ch> Channel number

Parameters:
<CorrData> Correction data

Example: `SOUR:POW:CORR:NIST:DEF 500000000, -10, 8000000000, -20`
Defines a non-ideal standard correction network, with 10 dB attenuation at 500 MHz, and 20 dB attenuation at 8 GHz.
`SENS1:CORR:POW:ACQ BWave, 1, Port, 1, OFF`
Perform a measurement receiver calibration at port 1, using port 1 as driving port and the previously defined correction data for the reflection standard.

SOURce<Ch>:POWer:CORRection:NISTandards:DELete

Deletes the correction buffer for non-ideal standards of channel <Ch>, created using `SOURce<Ch>:POWer:CORRection:NISTandards:DEFine` or `SOURce<Ch>:POWer:CORRection:NISTandards:LOAD`.

Until new correction data are specified, the firmware assumes ideal standards during subsequent measurement receiver power calibrations.

Suffix:
<Ch> Channel number

Usage: Event

SOURce<Ch>:POWer:CORRection:NISTandards:LOAD <TraceFile>, <Trace>

Loads an `snp` and applies one of its traces (in dB magnitude format) for correcting non-ideal standards during subsequent measurement receiver power calibrations. See "Calibration with non-ideal standards" on page 190.

The stimulus frequencies must "cover" the calibrated frequency range. If they do not coincide with the calibration sweep points, linear interpolation is used.

Suffix:
<Ch> Channel number

Setting parameters:
<TraceFile> Full path of the `snp` file, either absolute or relative to the current directory

<Trace> The S-parameter trace 'S<i><j>' to be used.

Example: `SOUR:POW:CORR:NIST:LOAD 'C:\Users\Public\Documents\Rohde-Schwarz\Vna\Traces\Test1.s1p' 'S11'`
Loads the file's S11 trace (in dB magnitude) into the internal buffer.
`SENS1:CORR:POW:ACQ BWave, 1, Port, 1, OFF`
Perform a measurement receiver calibration at port 1, using port 1 as driving port and the previously loaded correction data for the reflection standard.

Example: SOUR:POW:CORR:NIST:LOAD 'C:\Users\Public\Documents\Rohde-Schwarz\Vna\Traces\Test1.s2p' 'S12'
Loads the file's S21 trace (in dB magnitude) into the internal buffer.
SENS1:CORR:POW:ACQ BWave, 1, Port, 2, OFF
Perform a measurement receiver calibration at port 1, using port 2 as driving port and the previously loaded correction data for the Through standard.

Usage: Setting only

SOURce:POWer:CORRection:NREadings <NoCalSweeps>

Sets a limit for the number of calibration sweeps in a source power flatness cal. The command is valid for all channels, ports and external generators.

Equivalent command: SOURce:POWer:CORRection:COLLect:AVERage[:COUNt].

Parameters:
<NoCalSweeps> Number of readings
Range: 1 to 100
*RST: 2

Example: See SOURce<Ch>:POWer:CORRection[:ACQuire]

Manual operation: See "Flatness Cal – Max Iterations" on page 649

SOURce<Ch>:POWer:CORRection:OSOurces[:STATe] <Boolean>

Switches off all other sources during the calibration sweep for channel <Ch>.

Suffix:
<Ch> Calibrated channel number

Parameters:
<Boolean> ON - other sources not necessarily switched off.
OFF - other sources switched off during the calibration sweep.
*RST: ON

Example: See SOURce<Ch>:POWer:CORRection[:ACQuire]

Manual operation: See "Other Sources" on page 649

SOURce:POWer:CORRection:PMETer:ID <PowerMeter>

Selects an external power meter for the scalar source power calibration (flatness calibration).

Note: The command cannot be used unless a power meter is connected via GPIB bus, USB or LAN interface and configured in the "External Power Meters" dialog.

Parameters:

<PowerMeter> Number of external power meter. The parameters UP, DOWN, MIN, MAX are not available for this command.

Range: 1 to number of configured external power meters
*RST: The power meter selection is not changed by a reset of the analyzer.

Example: See `SOURce<Ch>:POWer<PhyPt>:CORRection:COLLect[:ACQuire]` on page 1644

Manual operation: See "Power Meter" on page 650

SOURce:POWer:CORRection:PPOWer <RecCalPower>

Defines the source power the R&S ZNB uses to perform the first calibration sweep of the source power calibration ("Reference Receiver Cal Power"). This predefined power value is ignored if the R&S ZNB is set to use the calculated port power result (`SOURce:POWer:CORRection:PSELect CPOW`).

Parameters:

<RecCalPower> Reference receiver cal power

Range: Depending on source power range of the analyzer and the power range of the external power meter.
*RST: 0 dBm
Default unit: dBm

Example: See `SOURce<Ch>:POWer:CORRection[:ACQuire]`.

Manual operation: See "Ref. Receiving Power" on page 355

SOURce:POWer:CORRection:PSELect <CalPower>

Defines how the R&S ZNB sets the power during the first calibration sweep of the source power calibration ("Reference Receiver Cal Power").

Parameters:

<CalPower> CPOWer | PPOWer

CPOWer
Calculated port power result

PPOWer
Predefined port power value, see `SOURce:POWer:CORRection:PPOWer`

*RST: CPOWer

Example: See `SOURce<Ch>:POWer:CORRection[:ACQuire]`.

Manual operation: See "Ref. Receiving Power" on page 355

SOURce<Ch>:POWer<PhyPt>:CORRection:STATe <Boolean>

Enables or disables the source power calibration for channel <Ch> and for port number <PhyPt>. The setting command is disabled unless a source power calibration for the analyzer port has been performed (SOURce<Ch>:POWer:CORRection[:ACQuire]). The query always returns a result.

To enable or disable a source power calibration for an external generator use SOURce<Ch>:POWer:CORRection:GENerator<Gen>[:STATe].

Suffix:
<Ch> Calibrated channel number

<PhyPt> Calibrated port number

Parameters:
<Boolean> Enables (ON) or disables (OFF) the source power calibration for port number <PhyPt>.

*RST: OFF

Example: See SOURce<Ch>:POWer:CORRection[:ACQuire].

Manual operation: See "Port Overview" on page 657

7.3.15.7 SOURce:POWer... (other)

The SOURce:POWer... commands define the power of the internal signal source.

SOURce<Ch>:POWer<PhyPt>:GENerator<Gen>:OFFSet ... 1523
SOURce<Ch>:POWer<PhyPt>:GENerator<Gen>:PERManent[:STATe] ... 1524
SOURce:POWer:GENerator:SDELay ... 1525
SOURce<Ch>:POWer:GENerator<Gen>:SLOPe ... 1525
SOURce<Ch>:POWer:GENerator<Gen>:STATe ... 1526
SOURce<Ch>:POWer<PhyPt>[:LEVel][:IMMediate][:AMPLitude] ... 1526
SOURce<Ch>:POWer<PhyPt>[:LEVel][:IMMediate]:OFFSet ... 1527
SOURce<Ch>:POWer<PhyPt>[:LEVel][:IMMediate]:SLOPe ... 1527
SOURce<Ch>:POWer<PhyPt>:PERManent[:STATe] ... 1528
SOURce:POWer:REDuce:SDELay ... 1528
SOURce<Ch>:POWer<PhyPt>:REDuce[:STATe] ... 1529
SOURce<Ch>:POWer<PhyPt>:STATe ... 1529
SOURce<Ch>:POWer<PhyPt>:STARt ... 1530
SOURce<Ch>:POWer<PhyPt>:STOP ... 1530
SOURce:POWer:SWEepend:MODE ... 1530
SOURce:POWer:SWEepend:SDELay ... 1531

SOURce<Ch>:POWer<PhyPt>:GENerator<Gen>:OFFSet <Offset>, <Mode>

Defines the power of an external generator or its power offset relative to the channel power (SOURce<Ch>:POWer<PhyPt>[:LEVel][:IMMediate][:AMPLitude]).

Suffix:
<Ch> Channel number

<PhyPt> Test port number of the analyzer. This suffix is ignored; the generator is selected via <Gen>.

<Gen> Generator number

Parameters:

<Offset> Port-specific power offset

Range: -300 dB to +300 dB (adjust to the generator's actual source power range and the test setup)
Increment: 0.01 dB (other values are rounded)
*RST: 0 dB
Default unit: dB

<Mode> ONLY | CPADd

ONLY - only the port-specific power is used; the channel power is ignored.
CPADd - the port-specific power is added as an offset to the channel power.

*RST: CPADd

Example:

```
*RST; SOUR:POW -6
```

Reset the instrument, activating a frequency sweep and set the internal source power (channel power) for the default channel 1 to -6 dBm.

```
SYST:COMM:RDEV:GEN1:DEF 'Ext. Gen. 1', 'SME02', 'gpib0', '21';
SYST:COMM:RDEV:GEN2:DEF 'Ext. Gen. 2', 'SME02', 'gpib0', '22'
```

Configure an R&S SME02 generator as external generator no. 1, assigning the name "Ext. Gen. 1" and a GPIB address 21. Configure a second R&S SME02 generator, assigning the name "Ext. Gen. 2" and a GPIB address 22.

```
SOUR:POW:GEN1:OFFS 6, ONLY; :SOUR:POW:GEN2:OFFS 6, CPAD
```

Set the source power of generator no. 1 to +6 dBm, the source power of generator no. 2 to 0 dBm.

Manual operation: See "Port Power Offset" on page 651

SOURce<Ch>:POWer<PhyPt>:GENerator<Gen>:PERManent[:STATe] <Boolean>

Defines whether the external generator is available as an external signal source in the test setup. External generators are always on for all partial measurements.

Suffix:

<Ch> Channel number

<PhyPt> Test port number of the analyzer. This suffix is ignored; the generator is selected via <Gen>.

<Gen> Generator number

Parameters:

<Boolean> ON - generator no. <PhyPt> is available.
OFF - generator is available only if a measurement result with a generator drive port is selected

*RST: OFF

Example: `SYST:COMM:RDEV:GEN1:DEF 'Ext. Gen. 1', 'SME02', 'gpib0', '21'`
Configure an R&S SME02 generator as external generator no. 1, assigning the name Ext. Gen. 1 and a GPIB address 21.
`SOUR:POW:GEN:PERM ON`
Activate the generator as an external signal source.

Manual operation: See "Source Gen" on page 687

SOURce:POWer:GENerator:SDELay <SettlingDelay>

If external generators are used in the measurement, this defines the minimum settling delay before the external generators are considered to be ready, i.e. the time between setting the generator to the requested frequency and power and continuing with the measurement.

This is a global setting.

Parameters:

<SettlingDelay> Settling delay per sweep point.
Factory default is 10 µs (= 10^{-5}s).

Default unit: s

Manual operation: See "Minimum Settling Delay" on page 821

SOURce<Ch>:POWer:GENerator<Gen>:SLOPe <Factor>

Defines a linear factor to modify the internal source power at generator <PhyPt> as a function of the stimulus frequency.

Suffix:

<Ch> Channel number

<Gen> Generator number

Parameters:

<Factor> Slope factor

Range: -40 dB/GHz to +40 dB/GHz. The resulting power range over the entire frequency sweep must be within the power range of the converter LO port; refer to the data sheet.
Increment: 0.1-dB/GHz
*RST: 0 dB/GHz
Default unit: DB/GHZ

Manual operation: See "Slope" on page 691

SOURce<Ch>:POWer:GENerator<Gen>:STATe <Boolean>

Turns an external generator numbered <Gen> on or off.

Suffix:
<Ch> Channel number

<Gen> Generator number

Parameters:
<Boolean> ON | OFF - generator is on or off
*RST: ON

Example: SYST:COMM:RDEV:GEN1:DEF 'Ext. Gen. 1', 'SME02', 'gpib0', '21'
Configure an R&S SME02 generator as external generator no. 1, assigning the name "Ext. Gen. 1" and a GPIB address 21.
SOUR:POW:GEN:STAT OFF
Turn the external generator off. In the "Port Configuration" dialog, "RF Off" is checked for generator no. 1.

Manual operation: See "Source RF" on page 687

SOURce<Ch>:POWer<PhyPt>[:LEVel][:IMMediate][:AMPLitude] <IntSourcePow>

Defines the power of the internal signal source (channel base power). The setting is valid for all sweep types, except power sweep.

Tip: Use SOURce<Ch>:POWer<PhyPt>:STARt and SOURce<Ch>:POWer<PhyPt>:STOP to define the sweep range for a power sweep.

Suffix:
<Ch> Channel number

<PhyPt> Test port number of the analyzer. This suffix is ignored because the selected channel base power applies to all source ports used in the active channel.
It is possible though to define a port-specific slope factor for the source power (SOURce<Ch>:POWer<PhyPt>[:LEVel][:IMMediate]:SLOPe) and a port-specific power offset (SOURce<Ch>:POWer<PhyPt>[:LEVel][:IMMediate]:OFFSet).

Parameters:
<IntSourcePow> Internal source power
Range: The usable power range is frequency-dependent; refer to the data sheet.
*RST: -10 dBm
Default unit: dBm

Example: `FUNC "XFR:POW:RAT B1, A2"`
Activate a frequency sweep and select the ratio B1/A2 as measured parameter for channel and trace no. 1.
`SOUR:POW -6`
Set the internal source power for channel 1 to -6 dBm.

Manual operation: See "Power" on page 340

SOURce<Ch>:POWer<PhyPt>[:LEVel][:IMMediate]:OFFSet <Offset>, <OffsetType>

Defines a port-specific source power or a power offset relative to the channel power (`SOURce<Ch>:POWer<PhyPt>[:LEVel][:IMMediate][:AMPLitude]`). An additional "Cal Power Offset" can be defined via `SOURce<Ch>:POWer<PhyPt>:CORRection:LEVel:OFFSet`.

Suffix:
<Ch> Channel number

<PhyPt> Test port number of the analyzer

Parameters:
<Offset> Port-specific power offset
Range: -300 dB to +300 dB (adjust to the analyzer's actual source power range and the test setup)
Increment: 0.01 dB (other values are rounded)
*RST: 0 dB
Default unit: dB

<OffsetType> ONLY | CPADd
ONLY - only the port-specific power is used; the channel power is ignored.
CPADd - the port-specific power is added as an offset to the channel power.
*RST: CPADd

Example: `*RST; SOUR:POW -6`
Reset the instrument, activating a frequency sweep and set the internal source power (channel power) for the default channel 1 to -6 dBm.
`SOUR:POW1:OFFS 6, ONLY; SOUR:POW2:OFFS 6, CPAD`
Replace the source power at port 1 by +6 dBm, the source power at port 2 by 0 dBm. The powers at the remaining ports (if available) are not affected.

Manual operation: See "Channel Base Power" on page 651

SOURce<Ch>:POWer<PhyPt>[:LEVel][:IMMediate]:SLOPe <Slope>

Defines a linear factor to modify the internal source power at port `<PhyPt>` as a function of the stimulus frequency.

Suffix:
<Ch> Channel number

<PhyPt> Test port number of the analyzer

Parameters:
<Slope> Port-specific slope factor

Range: -40 dB/GHz to +40 dB/GHz. The resulting power range over the entire frequency sweep must be within the power range of the analyzer; refer to the data sheet.
Increment: 0.1-dB/GHz
*RST: 0 dB/GHz
Default unit: DB/GHZ

Example: See `SOURce<Ch>:POWer<PhyPt>[:LEVel][:IMMediate][:AMPLitude]`

Manual operation: See "Slope" on page 691

SOURce<Ch>:POWer<PhyPt>:PERManent[:STATe] <Boolean>

Defines whether the source power is permanently on ("generator port").

Note:

If External switch matrices are part of the RF connection configuration, operation with more than one internal source is *not* supported. In this case only external generators can be used as permanent signal sources.

Suffix:
<Ch> Channel number

<PhyPt> Test port number of the analyzer

Parameters:
<Boolean> ON - power at port `<PhyPt>` is permanently on.
OFF - power is only on for the partial measurements that require the port as a drive port.

*RST: OFF

Example:
```
SOUR:POW2:STAT ON
```
Switch the RF power at port no. 2 on.
```
SOUR:POW2:PERM ON
```
Activate a permanent signal source.

Manual operation: See "Source Gen" on page 687

SOURce:POWer:REDuce:SDELay <SettlingDelay>

Deprecated; superseded by `SOURce:POWer:SWEepend:SDELay`

If power reduction at sweep end is enabled (`SOURce<Ch>:POWer<PhyPt>:REDuce[:STATe]` ON) this defines the settling delay at subsequent sweep starts, i.e. the time between power-up and sweep start when a new sweep is requested.

This is a global setting.

Parameters:

<SettlingDelay> Settling delay
Default unit: s

Example: See `SOURce:POWer:SWEepend:MODE` on page 1530

SOURce<Ch>:POWer<PhyPt>:REDuce[:STATe] <ReduceAtSweepEnd>

Deprecated; superseded by `SOURce:POWer:SWEepend:MODE`

Enables/disables power reduction at sweep end.

This is a global setting.

Suffix:

<Ch> Channel number. This suffix is ignored; the setting is valid for all channels.

<PhyPt> Physical port number. This suffix is ignored; the setting is valid for all ports.

Parameters:

<ReduceAtSweepEnd> **OFF** (default): at sweep end the output power of the sweep start is restored
ON: at sweep end the output power of the driving port is reduced as if the channel base power was set to its minimum possible value

Example: See `SOURce:POWer:SWEepend:MODE`.
`REDuce:STATe OFF` is equivalent to `SWEepend:MODE AUTO`
`REDuce:STATe ON` is equivalent to `SWEepend:MODE REDuce`

SOURce<Ch>:POWer<PhyPt>:STATe <Boolean>

Turns the RF source power at a specified test port on or off.

Suffix:

<Ch> Channel number

<PhyPt> Test port number of the analyzer

Parameters:

<Boolean> ON | OFF - turns the internal source power at the specified test port no. `<PhyPt>` on or off.
*RST: ON

Example: `*RST; SOUR:POW -6`
Set the internal source power for channel 1 and all test ports to -6 dBm.
`SOUR:POW2:STAT?`
Query whether the source power at test port 2 is on. The analyzer returns a `1`.

Manual operation: See "Source RF" on page 687

SOURce<Ch>:POWer<PhyPt>:STARt <StartPower>
SOURce<Ch>:POWer<PhyPt>:STOP <Value>

These commands define the start and stop powers for a power sweep. The values also define the left and right edges of a Cartesian diagram. A power sweep must be active (`[SENSe<Ch>:]SWEep:TYPE POWer`) to use these commands.

Suffix:
<Ch> Channel number

<PhyPt> Test port number of the analyzer. This suffix is ignored because the selected power sweep range applies to all source ports used in the active channel.

Parameters:
<Value> Start or stop power of the sweep.

Range: -150 dBm to +100 dBm. The usable power range is frequency-dependent; refer to the data sheet.
Increment: 0.01 dB
*RST: -25 dBm start power, 0 dBm stop power
Default unit: dBm

Example: `SWE:TYPE POW`
Activate a power sweep.
`SOUR:POW:STAR -6; STOP 10`
Select a power sweep range between -6 dBm and +10 dBm.

Manual operation: See "Start Frequency / Stop Frequency / Power CW Frequency" on page 341

Note: If the start power entered is greater than the current stop power (`SOURce<Ch>:POWer<PhyPt>:STOP`), the stop power is set to the start power plus the minimum power span (increment) of 0.01 dB.

If the stop power entered is smaller than the current start power (`SOURce<Ch>:POWer<PhyPt>:STARt`), the start power is set to the stop power minus the minimum power span of 0.01 dB.

SOURce:POWer:SWEepend:MODE <SweepEndModes>

Selects the power mode at sweep end.

This is a global setting.

Parameters:

<SweepEndModes> AUTO | REDuce | KEEP

AUTO: at sweep end restore the power at sweep start
REDuce: at sweep end reduce the output power as if the channel base power was set to its minimum possible value
KEEP: at sweep end keep the power at its current level

Example:

```
SOURce:POWer:SWEepend:MODE REDuce
```
Reduce power at sweep end.
```
SOURce:POWer:SWEepend:SDELay 0.01
```
Use a settling delay of 10 ms
```
SOURce:POWer:SWEepend:MODE AUTO
```
At sweep end, restore the power at sweep start

Manual operation: See "Power Mode at Sweep End" on page 822

SOURce:POWer:SWEepend:SDELay <SettlingDelay>

If sweep end mode REDuce or KEEP is active (see `SOURce:POWer:SWEepend:MODE`) this defines the settling delay at subsequent sweep starts, i.e. the time between power reset and sweep start when a new sweep is requested.

This is a global setting.

Parameters:

<SettlingDelay> Settling delay

Default unit: s

Manual operation: See "Settling Delay / Reset Delay" on page 823

7.3.15.8 SOURce... (other)

SOURce<Ch>:LPORt<LogPt> <PhysicalPort>[, <PhysicalPort>]

Assigns the logical port number <LogPt> either to the single-ended physical port <PhysicalPort1> or to a pair of physical ports <PhysicalPort1>,<PhysicalPort2>, at the same time defining them as balanced port.

Important:

- All required logical ports (balanced and single ended) must be created explicitly.
- A balanced port configuration generally introduces a new set of mixed-mode measured quantities. Therefore the traces must be redefined when a balanced port is created. To avoid any inconsistencies, the analyzer deletes all traces when `SOURce<Ch>:LPORt<LogPt>` is used.
- For the R&S ZNB, it is essential to complete the logical port assignment before defining other port properties such as common or differential mode impedances: whenever a logical port is created using `SOURce<Ch>:LPORt<LogPt>`, the parameters of all existing logical ports are reset. For the R&S ZNBT, this is not the case.
- A balanced port can only be created from ports in the same port group.

Suffix:

<Ch> Channel number.

<LogPt> Identifying number of the newly created balanced or single-ended logical port.
Must be between 1 and the number of physical ports (see `INSTrument:TPORt:COUNt?`). If unspecified the numeric suffix is set to 1.

Parameters:

<PhysicalPort> Number of the first physical port – or the only one for single-ended logical ports.

Range: 1 to number of physical ports

<PhysicalPort> Number of the second physical port (optional), forming a balanced port with the first physical port.
The port numbers must be different. Moreover, a physical port cannot be assigned to several logical ports.

Range: 1 to number of physical ports

Example:

```
*RST; :SOUR:LPOR1 1,2; :LPOR2 4; :LPOR3 3
```

Combine the physical ports 1 and 2 to the logical port 1 (balanced) and assign physical ports 4 and 3 to logical ports 2 and 3, respectively.

```
SOUR:LPOR1?
```

Query the physical ports assigned to logical port no. 1. The response is `1,2`.

```
SOUR:GRO 1,2
```

Define logical ports no. 1 and 2 as used (group 1), port 3 as unused.

```
SOUR:GRO?
```

Query the port group no. 1. The response is `1,2`.

```
SOUR:GRO:CLE ALL; :SOUR:GRO?
```

Dissolve all port groups and repeat the query. The response is `1,n`, where `n` denotes the number of available test ports.

```
SOUR:LPOR:CLE ALL
SOUR:LPOR1?
```

Dissolve all logical ports and repeat the query. The response is `1`: the logical port no. 1 is identical to the physical port no. 1.

Example:

```
SOUR:GRO:PORT 1,3
```

Define logical ports no. 1 and 3 as used (group 1), logical ports 2 and 4 as unused.

```
SOUR:GRO?
```

Query the port group no. 1. The response is `1,3`.

```
SOUR:GRO:CLE; :SOUR:GRO?
```

Dissolve port group 1 and repeat the query. The response is again `1,n` (all logical ports used).

Manual operation: See "Select Predefined Port" on page 297

SOURce<Ch>:LPORt<LogPt>:CLEar [<Scope>]

Dissolves balanced port <LogPt> or all logical ports.

Suffix:

<Ch> Channel number

<LogPt> Logical port number used to number balanced ports.
Range according to the current port configuration (SOURce<Ch>:LPORt<LogPt>). If an undefined balanced port number is used, the analyzer generates an error message.
Exception: Parameter ALL, see below.

Setting parameters:

<Scope> ALL

If ALL is specified, all logical ports are dissolved; the <log_port> suffix is ignored. If ALL is omitted, only the specified balanced port is dissolved.

Example: See SOURce<Ch>:LPORt<LogPt>

Usage: Setting only

Manual operation: See "Select Predefined Port" on page 297

7.3.16 STATus commands

The STATus:... commands control the status reporting system. Status registers are not reset by *RST; use *CLS for this purpose.

STATus:PRESet....1534
STATus:QUEStionable:CONDition?....1534
STATus:QUEStionable:INTegrity:CONDition?....1534
STATus:QUEStionable:INTegrity:HARDware:CONDition?....1534
STATus:QUEStionable:LIMit<Lev>:CONDition?....1534
STATus:QUEStionable:ENABle....1534
STATus:QUEStionable:INTegrity:ENABle....1534
STATus:QUEStionable:INTegrity:HARDware:ENABle....1534
STATus:QUEStionable:LIMit<Lev>:ENABle....1534
STATus:QUEStionable[:EVENt]?....1535
STATus:QUEStionable:INTegrity[:EVENt]?....1535
STATus:QUEStionable:INTegrity:HARDware[:EVENt]?....1535
STATus:QUEStionable:LIMit<Lev>[:EVENt]?....1535
STATus:QUEStionable:NTRansition....1535
STATus:QUEStionable:INTegrity:NTRansition....1535
STATus:QUEStionable:INTegrity:HARDware:NTRansition....1535
STATus:QUEStionable:LIMit<Lev>:NTRansition....1535
STATus:QUEStionable:PTRansition....1536
STATus:QUEStionable:INTegrity:PTRansition....1536
STATus:QUEStionable:INTegrity:HARDware:PTRansition....1536
STATus:QUEStionable:LIMit<Lev>:PTRansition....1536
STATus:QUEue[:NEXT]?....1536

STATus:PRESet

Configures the status reporting system such that device-dependent events are reported at a higher level.

The command affects only the transition filter registers, the ENABle registers, and queue enabling:

- The ENABle parts of the STATus:OPERation and STATus:QUEStionable... registers are set to all 1's.
- The PTRansition parts are set all 1's, the NTRansition parts are set to all 0's, so that only positive transitions in the CONDition part are recognized.

The status reporting system is also affected by other commands, see Section 6.5.5, "Reset values of the status reporting system", on page 903.

Example: STAT:PRES
Preset the status registers.

Usage: Event

STATus:QUEStionable:CONDition?
STATus:QUEStionable:INTegrity:CONDition?
STATus:QUEStionable:INTegrity:HARDware:CONDition?
STATus:QUEStionable:LIMit<Lev>:CONDition?

Returns the contents of the CONDition part of the QUEStionable... registers. Reading the CONDition registers is nondestructive.

Suffix:
<Lev> Selects one of the two QUEStionable:LIMit registers; see "STATus:QUEStionable:LIMit<1|2>" on page 897.

Example: STAT:QUES:LIMit:COND?
Query the CONDition part of the QUEStionable:LIMit1 register to retrieve the current status of the limit check.

Usage: Query only

STATus:QUEStionable:ENABle <BitPattern>
STATus:QUEStionable:INTegrity:ENABle <BitPattern>
STATus:QUEStionable:INTegrity:HARDware:ENABle <BitPattern>
STATus:QUEStionable:LIMit<Lev>:ENABle <BitPattern>

Sets the enable mask which allows true conditions in the EVENt part of the QUEStionable... registers to be reported in the summary bit. If a bit is 1 in the enable register and its associated event bit transitions to true, a positive transition will occur in the summary bit (e.g. bit 10 of the QUEStionable register for the LIMit1 register, bit 0 of the LIMit1 register for the LIMit2 register).

See also Section 6.5.1, "Overview of status registers", on page 891 and Section 6.5.5, "Reset values of the status reporting system", on page 903.

Suffix:
<Lev> Selects one of the two `QUEStionable:LIMit` registers; see "STATus:QUEStionable:LIMit<1|2>" on page 897.

Parameters:
<BitPattern> Range: 0 to 65535 (decimal representation)
*RST: n/a

Example: `STAT:QUES:LIM2:ENAB 6`
Set bits no. 1 and 2 of the `QUEStionable:LIMit2:ENABle` register

STATus:QUEStionable[:EVENt]?
STATus:QUEStionable:INTegrity[:EVENt]?
STATus:QUEStionable:INTegrity:HARDware[:EVENt]?
STATus:QUEStionable:LIMit<Lev>[:EVENt]?

These commands return the contents of the `EVENt` parts of the `QUEStionable`, `QUEStionable:INTegrity`, `QUEStionable:INTegrity:HARDware`, and `QUEStionable:LIMit<Lev>` status registers. Reading an `EVENt` register clears it.

Suffix:
<Lev> Selects one of the two `QUEStionable:LIMit` registers; see "STATus:QUEStionable:LIMit<1|2>" on page 897.

Example: `STAT:QUES:LIM1?`
Query the `EVENt` part of the `QUEStionable:LIMit1` register to check whether an event has occurred since the last reading.

Usage: Query only

STATus:QUEStionable:NTRansition <BitPattern>
STATus:QUEStionable:INTegrity:NTRansition <BitPattern>
STATus:QUEStionable:INTegrity:HARDware:NTRansition <BitPattern>
STATus:QUEStionable:LIMit<Lev>:NTRansition <BitPattern>

Sets the negative transition filters of the `QUEStionable...` status registers. If a bit is set, a 1 to 0 transition in the corresponding bit of the associated condition register causes a 1 to be written in the associated bit of the corresponding event register.

Suffix:
<Lev> Selects one of the two `QUEStionable:LIMit` registers; see "STATus:QUEStionable:LIMit<1|2>" on page 897.

Parameters:
<BitPattern> Range: 0 to 65535 (decimal representation)
*RST: n/a

Example: `STAT:QUES:LIM2:NTR 6`
Set bits no. 1 and 2 of the `QUEStionable:LIMit2:NTRansition` register

STATus:QUEStionable:PTRansition <BitPattern>
STATus:QUEStionable:INTegrity:PTRansition <BitPattern>
STATus:QUEStionable:INTegrity:HARDware:PTRansition <BitPattern>
STATus:QUEStionable:LIMit<Lev>:PTRansition <BitPattern>

Configures the positive transition filters of the `QUEStionable...` status registers. If a bit is set, a 0 to 1 transition in the corresponding bit of the associated condition register causes a 1 to be written in the associated bit of the corresponding event register.

See also Section 6.5.5, "Reset values of the status reporting system", on page 903.

Suffix:
<Lev> Selects one of the two `QUEStionable:LIMit` registers; see "STATus:QUEStionable:LIMit<1|2>" on page 897.

Parameters:
<BitPattern> Range: 0 to 65535 (decimal representation)
*RST: n/a

Example: `STAT:QUES:LIM2:PTR 6`
Set bits no. 1 and 2 of the `QUEStionable:LIMit2:PTRansition` register

STATus:QUEue[:NEXT]?

Queries and at the same time deletes the oldest entry in the error queue. Operation is identical to that of `SYSTem:ERRor[:NEXT]?`.

The entry consists of an error number and a short description of the error. Positive error numbers are instrument-dependent. Negative error numbers are reserved by the SCPI standard; see section Section 9, "Error messages and troubleshooting", on page 1687.

Example: `STAT:QUE?`
Query the oldest entry in the error queue. `0,"No error"` is returned if the error queue is empty.

Usage: Query only

7.3.17 SYSTem commands

The `SYSTem...` commands provide functions that are not related to instrument performance, such as functions for general housekeeping and functions related to global configurations.

7.3.17.1 SYSTem:COMMunicate...

The `SYSTem:COMMunicate...` commands provide remote control settings and configure remote (external) devices controlled by the R&S ZNB.

SYSTem:COMMunicate:AKAL:CONNection....1538
SYSTem:COMMunicate:AKAL:MMEMory[:STATe]....1538
SYSTem:COMMunicate:CODec....1539
SYSTem:COMMunicate:GPIB[:SELF]:ADDRess....1539
SYSTem:COMMunicate:GPIB[:SELF]:DCLear:SUPPress....1539
SYSTem:COMMunicate:GPIB[:SELF]:INIT:WAIT....1539
SYSTem:COMMunicate:GPIB[:SELF]:LPORt:ALIGn....1540
SYSTem:COMMunicate:GPIB[:SELF]:RTERminator....1540
SYSTem:COMMunicate:NET:HOSTname....1541
SYSTem:COMMunicate:RDEVice:AKAL:ADDRess....1541
SYSTem:COMMunicate:RDEVice:AKAL:ADDRess:ALL?....1541
SYSTem:COMMunicate:RDEVice:AKAL:ADDRess:SUBModule:ALL?....1541
SYSTem:COMMunicate:RDEVice:AKAL:CKIT....1542
SYSTem:COMMunicate:RDEVice:AKAL:CKIT:CATalog?....1543
SYSTem:COMMunicate:RDEVice:AKAL:CKIT:STANdard:CATalog?....1544
SYSTem:COMMunicate:RDEVice:AKAL:DATE?....1544
SYSTem:COMMunicate:RDEVice:AKAL:FRANge?....1544
SYSTem:COMMunicate:RDEVice:AKAL:PORTs?....1545
SYSTem:COMMunicate:RDEVice:AKAL:PREDuction[:STATe]....1545
SYSTem:COMMunicate:RDEVice:AKAL:SCAN?....1546
SYSTem:COMMunicate:RDEVice:AKAL:SDATa?....1546
SYSTem:COMMunicate:RDEVice:AKAL:TEMPerature?....1547
SYSTem:COMMunicate:RDEVice:AKAL:WARMup[:STATe]?....1547
SYSTem:COMMunicate:RDEVice:GDEVice:LAN:STATe....1547
SYSTem:COMMunicate:RDEVice:GENerator<Gen>:CATalog?....1547
SYSTem:COMMunicate:RDEVice:GENerator:COUNt?....1548
SYSTem:COMMunicate:RDEVice:GENerator<Gen>:DEFine....1548
SYSTem:COMMunicate:RDEVice:GENerator:DELete....1549
SYSTem:COMMunicate:RDEVice:GENerator:SCAN?....1550
SYSTem:COMMunicate:RDEVice:GENerator<Gen>:SEPMode....1550
SYSTem:COMMunicate:RDEVice:GENerator<Gen>:SEPower....1550
SYSTem:COMMunicate:RDEVice:PMETer<Pmtr>:AZERo....1551
SYSTem:COMMunicate:RDEVice:PMETer:CATalog?....1551
SYSTem:COMMunicate:RDEVice:PMETer:CONFigure:AUTO[:STATe]....1551
SYSTem:COMMunicate:RDEVice:PMETer:COUNt?....1552
SYSTem:COMMunicate:RDEVice:PMETer<Pmtr>:DEFine....1552
SYSTem:COMMunicate:RDEVice:PMETer:DELete....1553
SYSTem:COMMunicate:RDEVice:PMETer:SCAN?....1553
SYSTem:COMMunicate:RDEVice:PMETer<Pmtr>:SPCorrection[:STATe]....1553
SYSTem:COMMunicate:RDEVice:SMATrix<Matr>:CATalog?....1554
SYSTem:COMMunicate:RDEVice:SMATrix:CONFigure:ABORt....1554
SYSTem:COMMunicate:RDEVice:SMATrix:CONFigure:END....1554
SYSTem:COMMunicate:RDEVice:SMATrix:CONFigure:STARt....1555
SYSTem:COMMunicate:RDEVice:SMATrix:CONFigure:TVNA....1556
SYSTem:COMMunicate:RDEVice:SMATrix<Matr>:CONFigure:MLVNa....1557
SYSTem:COMMunicate:RDEVice:SMATrix<Matr>:CONFigure:MLTest....1557
SYSTem:COMMunicate:RDEVice:SMATrix<Matr>:CONFigure:MTESt....1558
SYSTem:COMMunicate:RDEVice:SMATrix<Matr>:CONFigure:MVNA....1558
SYSTem:COMMunicate:RDEVice:SMATrix:COUNt?....1559
SYSTem:COMMunicate:RDEVice:SMATrix<Matr>:DEFine....1559

SYSTem:COMMunicate:RDEVice:SMATrix:DELete 1560
SYSTem:COMMunicate:RDEVice:SMATrix<Matr>:RELays:SWITch:COUNt? 1561
SYSTem:COMMunicate:RDEVice:SMATrix:SCAN? 1561
SYSTem:COMMunicate:RDEVice:SMATrix<Matr>:SERial? 1561

SYSTem:COMMunicate:AKAL:CONNection <CalStandard>[, <Port>[, <SecondPort>[, <CalkitIndex>]]]

Connects the selected calibration standard to one or two ports of the active calibration unit (see `SYSTem:COMMunicate:RDEVice:AKAL:ADDRess`).

Setting parameters:

<CalStandard> THRough | OPEN | SHORt | MATCh | NONE

One- or two-port standard connected to the port or ports.
NONE switches the calibration unit to the "non-calibrating" state. Inline calibration units R&S ZN-Z3x switch from the calibration to the measurement state.

<Port> Port number of the calibration unit, for one and two-port standards.

<SecondPort> Second port number of the calibration unit.
For two-port standards (THRough) this parameter must be provided, for one-port standards (OPEN | SHORt | MATCh) it must be omitted.

<CalkitIndex> For calibration units that are equipped with more than one calibration kit, this parameter indicates the 1-based index of the cal kit to be used (mandatory!).
For other calibration units, this parameter is optional, but if it is provided, it must be set to 1.

Example: `SYST:COMM:AKAL:CONN THR, 1, 2`
Connect a through standard between ports 1 and 2 of the cal unit.

Usage: Setting only

Manual operation: See "Apply/Cancel" on page 576

SYSTem:COMMunicate:AKAL:MMEMory[:STATe] <Boolean>

Shows or hides the memory of the active calibration unit (see `SYSTem:COMMunicate:RDEVice:AKAL:ADDRess`).

Setting parameters:

<Boolean> ON - memory is shown in a separate drive.
OFF - memory is not shown.

*RST: OFF

Example: `SYST:COMM:AKAL:MMEM ON`
Show the memory of the active calibration unit.

Usage: Setting only

Manual operation: See "Apply/Cancel" on page 576

SYSTem:COMMunicate:CODec <Codec>

Selects the character encoding used at the remote interface. The selected encoding applies to directory and file names, CalKit names, CalUnit characterizations and display titles.

Parameters:

<Codec> ASCii | UTF8 | SJIS

ASCii: 8-bit ANSI (default)
UTF8: UTF-8
SJIS: Shift JIS

Manual operation: See "Remote Encoding" on page 810

SYSTem:COMMunicate:GPIB[:SELF]:ADDRess <GBIBAddress>

Sets the GPIB address of the analyzer.

Parameters:

<GBIBAddress> GPIB address (integer number)

Range: 0 to 30

Example: `SYST:COMM:GPIB:ADDR 10`
Set the GPIB address to 10.
`*RST; :SYST:COMM:GPIB:ADDR?`
After a reset, the address is maintained (the response is 10).

Manual operation: See "GPIB Address" on page 843

SYSTem:COMMunicate:GPIB[:SELF]:DCLear:SUPPress <Boolean>

Suppresses/unsuppresses of Device Clear GPIB interface messages (DCL, SDC).

Parameters:

<Boolean>

Manual operation: See "GPIB Address" on page 843

SYSTem:COMMunicate:GPIB[:SELF]:INIT:WAIT <Boolean>

Determines/queries the execution behavior of `INITiate[:IMMediate]` commands (see Section 7.3.8, "INITiate commands", on page 1247).

If set to ON, an automatic `*WAI` is added (see Section 7.2, "Common commands", on page 913), i.e. the commands execute synchronously.

Parameters:

<Boolean> *RST: OFF

Manual operation: See "Advanced ..." on page 844

SYSTem:COMMunicate:GPIB[:SELF]:LPORt:ALIGn <Boolean>

Configures/queries the logical port creation.

If set to ON, logical ports are aligned and must be set from low to high port. If set to OFF, new ports can be created freely, like in manual operation. See the example below.

Parameters:

<Boolean> *RST: ON

Example: A `*RST` on a 4-port analyzer restores its default logical port assignment L1↔P1, L2↔P2, L3↔P3, L4↔P4.
With `ALIGn=ON` (default), `SOUR:LPOR1 1,2` generates aligned logical ports L1↔P1&P2, L2↔P3, L3↔P4, i.e. the other logical ports are renumbered in ascending order.
With `ALIGn=OFF`, the result is L1↔P1&P2, L3↔P3, L4↔P4 and L2 does not exist.

Manual operation: See "Advanced ..." on page 844

SYSTem:COMMunicate:GPIB[:SELF]:RTERminator <Terminator>

Sets the receive terminator of the analyzer. The receive terminator indicates the end of a command or a data block.

The receive terminator setting is relevant if block data is transferred to the analyzer (`FORMat[:DATA] REAL`). In the default setting LFEoi, the analyzer recognizes an LF character sequence with or without the EOI control bus message as a receive terminator. An accidental LF in a data block can be recognized as a terminator and cause an interruption of the data transfer.

The EOI setting is especially important if commands are transferred in block data format, because it ensures that the parser for command decoding is activated by the terminator only after the command has been completely transferred. Readout of binary data does not require a change of the receive terminator.

Note: since firmware V2.80 this command raises an error if no GPIB interface (option R&S ZNB-B10) is available.

Parameters:

<Terminator> LFEoi | EOI

LFEoi
A line feed character sequence with or without EOI is recognized as receive terminator

EOI
Only EOI is recognized as receive terminator

Example: `SYST:COMM:GPIB:RTER EOI`
Set the terminator to EOI.

SYSTem:COMMunicate:NET:HOSTname <HostName>

Sets or gets the host name of the instrument.

The default host name is

- ZNB-<serial no.> for R&S ZNB
- ZNBT-<serial no.> for R&S ZNBT

Parameters:

<HostName> Host name

*RST: A preset of the instrument doesn't change the host name.

SYSTem:COMMunicate:RDEVice:AKAL:ADDRess <Address>

Selects one of the USB-connected calibration units for calibration (see commands `SENSe<Ch>:CORRection:COLLect:AUTO...`). This command is not necessary if only one cal unit is connected.

Parameters:

<Address> Name (USB address) of a connected calibration unit (string variable). The names of all connected cal units can be queried using `SYSTem:COMMunicate:RDEVice:AKAL:ADDRess:ALL?`.

Example:

```
SYST:COMM:RDEV:AKAL:ADDR:ALL?
```

Query the names of all connected calibration units.

```
SYST:COMM:RDEV:AKAL:ADDR 'ZV-Z52::1234'
```

Select the cal unit named 'ZV-Z52::1234' for calibration.

```
CORR:COLL:AUTO '', 1, 2, 4
```

Perform an automatic 3-port TOSM calibration at test ports 1, 2, and 4 using the calibration unit's default calibration kit file and arbitrary test ports of the cal unit.

Manual operation: See "Cal Unit" on page 568

SYSTem:COMMunicate:RDEVice:AKAL:ADDRess:ALL?

Queries the names (USB addresses) of all connected calibration units.

Example: See `SYSTem:COMMunicate:RDEVice:AKAL:ADDRess`

Usage: Query only

Manual operation: See "Cal Unit" on page 568

SYSTem:COMMunicate:RDEVice:AKAL:ADDRess:SUBModule:ALL?

If the calibration unit selected using `SYSTem:COMMunicate:RDEVice:AKAL:ADDRess` is an inline calibration system (ICS), this query returns the addresses of the connected ICUs. Otherwise it returns the empty string.

Example:
```
:SYSTem:COMMunicate:RDEVice:AKAL:
ADDRess 'ZN-Z30-02::100010'
```
Selects an ICS via its Inline Calibration Controller (ICC) R&S ZN-Z30 (serial number 100010).
```
SYSTem:COMMunicate:RDEVice:AKAL:ADDRess:
SUBModule:ALL?
```
Returns something like:
```
'[1]ZN-Z33-02::101325,[2]ZN-Z33-02::
101325,[3]ZN-Z33-02::101339'
```
i.e. there are three ICUs of type R&S ZN-Z32 variant 02 connected to the ICC. For correct port assignment during calibration, the ICU with serial number 101325 is considered port 1, 101325 port 2, and 101339 port 3 of the ICS.

Usage: Query only

Manual operation: See "ZN-Z32/ZN-Z33" on page 863

SYSTem:COMMunicate:RDEVice:AKAL:CKIT <Characterization>

If the calibration unit selected using `SYSTem:COMMunicate:RDEVice:AKAL:ADDRess` is an inline calibration unit (ICU), this command sets/queries the characterization to be used for calibrations with this ICU.

Parameters:

<Characterization> Name of a characterization of the related ICU

Example:
```
// Select the ICS
SYSTem:COMMunicate:RDEVice:AKAL:ADDRess 'ZN-Z30-02::100010'
// Query for the connected ICUs
SYSTem:COMMunicate:RDEVice:AKAL:ADDRess:SUBModule:ALL?
// '[1]ZN-Z33-02::101325,[2]ZN-Z33-02::101336,[3]ZN-Z33-02::101339'
// Activate ICU[1]
SYSTem:COMMunicate:RDEVice:AKAL:ADDRess 'ZN-Z33-02::101336'
// Query the available characterizations
:SYSTem:COMMunicate:RDEVice:AKAL:CKIT:CATalog?
// 'Factory,afterAwhile'
// Select the non-factory characterization
:SYSTem:COMMunicate:RDEVice:AKAL:CKIT 'afterAwhile'
// Query the active characterization
:SYSTem:COMMunicate:RDEVice:AKAL:CKIT?
// 'ZN-Z33-02::101336=afterAwhile'
```

Manual operation: See "Characterization" on page 568

SYSTem:COMMunicate:RDEVice:AKAL:CKIT:CATalog?

Queries all characterizations (cal kit files) which are stored on the connected calibration unit, either on the calibration unit's internal (flash) memory or on an SD card inserted at the calibration unit (prefix: "SD"). A possible response is `'Factory,ZN-Z51_custom,Throughs,SD:test'`. The factory characterization is always available; an empty string denotes that no calibration unit is connected.

If several cal units are USB-connected to the analyzer, the command queries the cal unit selected via `SYSTem:COMMunicate:RDEVice:AKAL:ADDRess`.

Example:

```
SYST:COMM:RDEV:AKAL:ADDR:ALL?
```
Query the names of all connected calibration units.
```
SYST:COMM:RDEV:AKAL:ADDR 'ZN-Z51::1234'
```
Select the cal unit named 'ZN-Z51::1234'.
```
SYSTem:COMMunicate:RDEVice:AKAL:CKIT:CATalog?
```
Query all characterizations stored on the connected calibration unit.
```
SYSTem:COMMunicate:RDEVice:AKAL:SDATa?
'My_calkit', OPEN, S11, 1
```
Query the characterization data for the characterization named 'My_calkit' and an open standard (one-port, port restriction). A characterization with the queried properties must be available on the cal unit.
```
SYSTem:COMMunicate:RDEVice:AKAL:WARMup?
```
Query the warmup status of the calibration unit.
```
SYSTem:COMMunicate:RDEVice:AKAL:DATE? 'Factory'
```
Query the creation date of the factory calibration.
```
SYSTem:COMMunicate:RDEVice:AKAL:FRANge?
'Factory'
```
Query the frequency range of the factory calibration.
```
SYSTem:COMMunicate:RDEVice:AKAL:PORTs?
'Factory'
```
Query the port assignment of the factory calibration.
```
SYSTem:COMMunicate:RDEVice:AKAL:CKIT:STANdard:
CATalog? 'Factory'
```
Query the standards of the factory calibration. Possible response:
`'MOP(P1),MSH(P1),MMMT(P1),MOP(P2),MSH(P2),MMMT(P2)'` – denotes an Open (m), Short (m) and Match (m) standard at each of the ports 1 and 2.

Usage: Query only

Manual operation: See "Characterization" on page 568

SYSTem:COMMunicate:RDEVice:AKAL:CKIT:STANdard:CATalog? <CalKitName>

Queries all calibration standards of a characterization (calkit file) that is stored on a connected calibration unit, either on the calibration unit's internal (flash) memory or on an SD card inserted at the calibration unit (if available). The query `SYSTem:COMMunicate:RDEVice:AKAL:CKIT:CATalog?` returns the names of the cal kit files. The factory characterization is always available.

Query parameters:

<CalKitName> Name of the calkit file, string parameter.
`'Factory'` denotes the factory characterization, the prefix "SD:" indicates that the characterization is stored on an SD card inserted at the calibration unit.

Example: See `SYSTem:COMMunicate:RDEVice:AKAL:CKIT:CATalog?`

Usage: Query only

Manual operation: See "Characterization" on page 568

SYSTem:COMMunicate:RDEVice:AKAL:DATE? <CalKitName>

Queries the creation date and time of the cal unit characterization (calkit file) <CalKitName>. A possible response is `'Friday, May 26, 2011, 10:13:40'`. An empty string is returned if no calibration unit is connected.

If several cal units are USB-connected to the analyzer, the command queries the cal unit selected via `SYSTem:COMMunicate:RDEVice:AKAL:ADDRess`.

Query parameters:

CalKitName String parameter containing the name of a cal unit characterization (calkit file).
`'Factory'` denotes the factory characterization; an empty string `' '` refers to the last referenced characterization; the prefix "SD:" indicates that the characterization is stored on an SD card inserted at the calibration unit.

Example: See `SYSTem:COMMunicate:RDEVice:AKAL:CKIT:CATalog?`

Usage: Query only

Manual operation: See "Characterization" on page 568

SYSTem:COMMunicate:RDEVice:AKAL:FRANge? <CalKitName>

Queries the frequency range of the cal unit characterization (calkit file) <CalKitName>. The response consists of the start and stop frequencies in Hz, separated by a comma. `0,0` is returned if no calibration unit is connected.

If several cal units are USB-connected to the analyzer, the command queries the cal unit selected via `SYSTem:COMMunicate:RDEVice:AKAL:ADDRess`.

Query parameters:

<CalKitName> String parameter containing the name of a cal unit characterization (calkit file).
`'Factory'` denotes the factory characterization; an empty string `' '` denotes the last referenced characterization; the prefix "SD:" indicates that the characterization is stored on an SD card inserted at the calibration unit.

Example: See `SYSTem:COMMunicate:RDEVice:AKAL:CKIT:CATalog?`

Usage: Query only

Manual operation: See "Characterization" on page 568

SYSTem:COMMunicate:RDEVice:AKAL:PORTs? <CalKitName>

Queries the number of ports of a cal unit characterization (calkit file) <CalKitName>, the assigned connector types, and their gender. A possible response for a two-port calibration is `'1,N 50 Ohm,MALE,2,N 50 Ohm,MALE'`. An empty string is returned if no calibration unit is connected.

If several cal units are USB-connected to the analyzer, the command queries the cal unit selected via `SYSTem:COMMunicate:RDEVice:AKAL:ADDRess`.

Query parameters:

<CalKitName> String parameter containing the name of a cal unit characterization (calkit file).
`'Factory'` denotes the factory characterization; an empty string `' '` denotes the last referenced characterization; the prefix "SD:" indicates that the characterization is stored on an SD card inserted at the calibration unit.

Example: See `SYSTem:COMMunicate:RDEVice:AKAL:CKIT:CATalog?`

Usage: Query only

Manual operation: See "Characterization" on page 568

SYSTem:COMMunicate:RDEVice:AKAL:PREDuction[:STATe] <Boolean>

Enables or disables automatic power reduction at all test ports while an automatic calibration is active.

Parameters:

<Boolean> Power reduction enabled or disabled.

*RST: ON

Example: `SYSTem:COMMunicate:RDEVice:AKAL:PREDuction OFF`
Disable automatic power reduction.

Manual operation: See "Auto Power Reduction for Cal Unit" on page 813

SYSTem:COMMunicate:RDEVice:AKAL:SCAN?

Scans for calibration units.

Usage: Query only

SYSTem:COMMunicate:RDEVice:AKAL:SDATa? <CalKitName>, <Type>, <SParameter>[, <FirstPort>, <SecondPort>]

Reads the calibration data for a particular standard from a cal unit characterization (calkit file). If more than one calibration unit is connected, the related one must be selected using `SYSTem:COMMunicate:RDEVice:AKAL:ADDRess`.

Query parameters:

<CalKitName> String parameter containing the name of a cal unit characterization (calkit file).
`'Factory'` denotes the factory characterization; an empty string `' '` denotes the last referenced characterization; the prefix "SD:" indicates that the characterization is stored on an SD card inserted at the calibration unit.

<Type> THRough | OPEN | SHORt | MATCh | MMTHrough | MFTHrough | FFTHrough | MOPen | FOPen | MSHort | FSHort | MMTCh | FMTCh

Standard types; for a description refer to table Standard types and their parameters. The standard types of a particular characterization can be queried via `SYSTem:COMMunicate:RDEVice:AKAL:CKIT:STANdard:CATalog?`.
The factory characterization usually does not contain data for a through standard; therefore a query of the type `SYSTem:COMMunicate:RDEVice:AKAL:SDATa? 'Factory', THR, S11, 1, 2` results in an error message.

<SParameter> S11 | S12 | S21 | S22

S-parameter of the standard, use `S11` for one-port standards, one of the four for 2-port standards.

<FirstPort> First port number (sufficient for one-port standards). Port numbers can be omitted if the cal kit data is valid for all ports; see `MMEMory:LOAD:CKIT:SDATa`.

<SecondPort> Second port number, for two-port standards. Port numbers can be omitted if the cal kit data is valid for all ports.

Example: See `SYSTem:COMMunicate:RDEVice:AKAL:CKIT:CATalog?`

Usage: Query only

Manual operation: See "Characterization" on page 568

SYSTem:COMMunicate:RDEVice:AKAL:TEMPerature?

Queries the current temperature of the active calibration unit (selected via `SYSTem:COMMunicate:RDEVice:AKAL:ADDRess`, if more than one is connected).

Usage: Query only

Manual operation: See "Calibration Unit" on page 644

SYSTem:COMMunicate:RDEVice:AKAL:WARMup[:STATe]?

Queries the warmup state of the connected calibration unit R&S ZV_Z5x. If several cal units are USB-connected to the analyzer, the command queries the cal unit selected via `SYSTem:COMMunicate:RDEVice:AKAL:ADDRess`.

Possible responses are `1` (true, if the calibration unit has been connected for a sufficient time to rach its operating temperature) or `0` (false). `0` is also returned if no calibration unit is connected.

Example: See `SYSTem:COMMunicate:RDEVice:AKAL:CKIT:CATalog?`

Usage: Query only

Manual operation: See "Characterization" on page 568

SYSTem:COMMunicate:RDEVice:GDEVice:LAN:STATe <Boolean>

Activates or deactivates LAN detection of external devices.

Parameters:

<Boolean>
ON (1)
LAN detection activated
OFF (0)
LAN detection deactivated
*RST: OFF (0)

Manual operation: See "LAN Detection" on page 853

SYSTem:COMMunicate:RDEVice:GENerator<Gen>:CATalog?

Queries the numbers of all configured external generators. The response is a string containing a comma-separated list of generator numbers.

Suffix:

<Gen> Number of the configured generator. This suffix is ignored; the command affects all generators.

Example:
```
SYST:COMM:RDEV:GEN1:DEF 'Ext. Gen. 1', 'SME02',
'gpib0',  '21'
SYST:COMM:RDEV:GEN2:DEF 'Ext. Gen. 2', 'SME02',
'gpib0',  '22'
```
Configure two R&S SME02 generators as external generators no. 1 and 2, assigning different names and GPIB addresses.
```
SYST:COMM:RDEV:GEN1:CAT?
```
Query the generator numbers. The response is '1, 2'.
```
SYST:COMM:RDEV:GEN1:COUN?
```
Query the number of configured generators. The response is 2.

Usage: Query only

Manual operation: See "Configured Devices" on page 852

SYSTem:COMMunicate:RDEVice:GENerator:COUNt?

Queries the number of configured external generators. The response is an integer number of generators.

Example: See `SYSTem:COMMunicate:RDEVice:GENerator<Gen>:CATalog?`

Usage: Query only

SYSTem:COMMunicate:RDEVice:GENerator<Gen>:DEFine <GenName>, <Driver>, <Interface>, <Address>[, <FastSweep>[, <ExtReference>]]

Configures an external generator and adds it to the list of available generators.

Suffix:

<Gen> Number of the configured generator. Generators must be numbered in ascending order, starting with 1. If a number is reused, the previous generator configuration is overwritten. Generators can be assigned several times so that the number of configured generators is practically unlimited.

Parameters:

<GenName> Name of the external generator (string parameter). An empty string means that no particular name is assigned to the generator.

<Driver> Generator type (string parameter). The generator type is identical with the name of the generator driver file (`*.gen`) stored in the resources\extdev subdirectory of the analyzer's program directory. Type the driver filename as shown in the "Add External Generator" dialog, i.e. without the file extension "`.gen`" (example: use '`sme02`', if the corresponding driver filename is `sme02.gen`).
Alternative: The '`<driver>`' string may also contain the generator driver filename with its complete path, e.g. '`C:\Program Files\Rohde-Schwarz\Vector Network Analyzer\ZNB\resources\extdev\sme02.gen`'.

<Interface> Interface type (string parameter): `'HiSLIP'`, `'VXI-11'`, `'SOCKET'`, `'USB-VISA'`, `'GPIB0'`, ..., `'GPIB9'`, `'other'` ...

<Address> Interface address (string parameter), depending on the interface type. See Section 5.17.5.2, "External Generators dialog", on page 851.

<FastSweep> ON | OFF

Optional boolean parameter, enables or disables the fast sweep mode.

<ExtReference> ON | OFF

Optional Boolean parameter

ON
External reference frequency

OFF
Internal reference frequency

Example:
`SYST:COMM:RDEV:GEN1:DEF 'Ext. Gen. 1', 'SME02', 'gpib0', '21'`
Configure an R&S SME02 generator as external generator no. 1, assigning the name "Ext. Gen. 1" and a GPIB address 21.
`SYST:COMM:RDEV:GEN1:DEF?`
Query the generator configuration. The response `'Ext. Gen. 1','SME02','GPIB0','21',OFF, OFF` indicates that the fast sweep mode is disabled and that the analyzer is set to internal reference frequency.
`SYST:COMM:RDEV:GEN1:DEL`
Clear the generator configuration table.
`SYST:COMM:RDEV:GEN1:DEF?`
Query the generator configuration. The analyzer returns an error message because the generator no. 1 is no longer configured.

Manual operation: See "Configured Devices" on page 852

SYSTem:COMMunicate:RDEVice:GENerator:DELete

Clears the configuration table for external generators.

Example: See SYSTem:COMMunicate:RDEVice:GENerator<Gen>:DEFine

Usage: Event

Manual operation: See "Configured Devices" on page 852

SYSTem:COMMunicate:RDEVice:GENerator:SCAN?

Scans for external generators

Usage: Query only

Manual operation: See "Scan Instruments" on page 853

SYSTem:COMMunicate:RDEVice:GENerator<Gen>:SEPMode <Type>

Defines the external generator power control after the sweep end.

Suffix:
<Gen> Generator number

Parameters:
<Type> OFF | LOW | KEEP | USER

OFF – power is switched off
LOW – switch to min. power
KEEP – keep power at the value selected via SOURce<Ch>:POWer<PhyPt>:GENerator<Gen>:OFFSet
USER – set to end power selected via SYSTem:COMMunicate:RDEVice:GENerator<Gen>:SEPower

Example: See SYSTem:COMMunicate:RDEVice:GENerator<Gen>:SEPower

Manual operation: See "Configured Devices" on page 852

SYSTem:COMMunicate:RDEVice:GENerator<Gen>:SEPower <Power>

Defines the external generator power value after the end of a sweep. The setting is relevant if the generator is switched to a user-selected power; see example.

Suffix:
<Gen> Generator number

Parameters:
<Power> End power

Range: -145 dBm to +30 dBm
*RST: 0 dBm
Default unit: dBm

Example: `SYSTem:COMMunicate:RDEVice:GENerator:SEPMode USER`
Switch the generator to a user-selected power value after the sweep is terminated.
`SYSTem:COMMunicate:RDEVice:GENerator:SEPower -10`
Define a end power of -10 dBm.

Manual operation: See "Configured Devices" on page 852

SYSTem:COMMunicate:RDEVice:PMETer<Pmtr>:AZERo

Starts auto zeroing of the external power meter.

Suffix:
<Pmtr> Number of the configured power meter. Power meters must be numbered in ascending order, starting with 1. If a number is re-used, the previous power meter configuration is overwritten. Power meters can be assigned several times so that the number of configured power meters is practically unlimited.

Usage: Event

Manual operation: See "Auto Zero" on page 319

SYSTem:COMMunicate:RDEVice:PMETer:CATalog?

Queries the numbers of all configured external power meters. The response is a string containing a comma-separated list of power meter numbers.

Example: `SYST:COMM:RDEV:PMET1:DEF 'USB Power Meter 1', 'NRP-Z55', 'usb', '100045'`
`SYST:COMM:RDEV:PMET2:DEF 'USB Power Meter 2', 'NRP-Z55', 'usb', '100046'`
Configure two R&S NRP power meters as external power meter no. 1 and 2, assigning names and serial numbers.
`SYST:COMM:RDEV:PMET:CAT?`
Query the power meter numbers. The response is '1, 2'.
`SYST:COMM:RDEV:PMET:COUN?`
Query the number of configured power meters. The response is 2.

Usage: Query only

Manual operation: See "Configured Devices" on page 846

SYSTem:COMMunicate:RDEVice:PMETer:CONFigure:AUTO[:STATe] <Boolean>

Enables or disables auto-configuration of R&S NRPxxS/SN power sensors. If the function is enabled, the analyzer automatically configures all power sensors detected at any of the USB ports as Pmtr 1, Pmtr 2, ...

Parameters:

<Boolean> ON | OFF – enable or disable auto-configuration.

Example: See `SYSTem:COMMunicate:RDEVice:PMETer<Pmtr>:DEFine`

Manual operation: See "Auto Config" on page 849

SYSTem:COMMunicate:RDEVice:PMETer:COUNt?

Queries the number of configured external power meters. The result is an integer number of power meters.

Example: See `SYSTem:COMMunicate:RDEVice:PMETer:CATalog?`

Usage: Query only

Manual operation: See "Configured Devices" on page 846

SYSTem:COMMunicate:RDEVice:PMETer<Pmtr>:DEFine <PmtrName>, <Driver>, <Interface>, <Address>[, <SensorId>]

Configures an external power meter and adds it to the list of available power meters.

Suffix:

<Pmtr> Number of the configured power meter. Power meters must be numbered in ascending order, starting with 1. If a number is reused, the previous power meter configuration is overwritten. Power meters can be assigned several times so that the number of configured power meters is practically unlimited.

Parameters:

<PmtrName> Name of the external power meter (string parameter). An empty string means that no particular name is assigned to the power meter.

<Driver> Power meter type (string parameter). The power meter type is identical with the name of the power meter driver file (`*.pwm`) stored in the `resources\extdev` subdirectory of the analyzer's program directory. Type the driver file name as shown in the "Add External Powermeter" dialog, i.e. without the file extension "`.pwm`" (example: use '`NRVD`', if the corresponding driver file name is `NRVD.pwm`).
Alternative: The '<Driver>' string can also contain the power meter driver file name with its complete path, e.g. '`C:\Program Files\Rohde-Schwarz\Vector Network Analyzer\ZNB\resources\extdev\nrvd.pwm`'.

<Interface> Interface type (string parameter): `'VXI-11'`, `'socket'`, `'GPIB0'`, ..., `'GPIB9'`, `'other'`

<Address> Interface address (string parameter), depending on the interface type. See Table 5-11

<SensorId> This optional string parameter is used for power meters that can control several power sensors, such as the R&S NRP2. By specifying the suitable <SensorId> you can address the related power sensor.
The R&S NRP2, for example, has four power sensor connectors, Sensor A to Sensor D. By specifying `'A'`, `'B'`, `'C'` or `'D'`, you can take control over the power sensor that is attached to this connector.

Example: `SYST:COMM:RDEV:PMET:CONF:AUTO OFF`
Disables "Auto Config NRP-Zxx" (if it was enabled previously).
`SYST:COMM:RDEV:PMET1:DEF 'USB Power Meter', 'NRP-Z55', 'usb', '100045'`
Configure an R&S NRP power meter as external power meter no. 1, assigning the name "USB Power Meter" and a serial number 100045.
`SYST:COMM:RDEV:PMET:DEL`
Clear the power meter configuration table.
`SYST:COMM:RDEV:PMET:DEF?`
Query the power meter configuration. The analyzer returns an error message because the power meter no. 1 is no longer configured.

Manual operation: See "Configured Devices" on page 846

SYSTem:COMMunicate:RDEVice:PMETer:DELete

Clears the configuration table for external power meters.

Example: See `SYSTem:COMMunicate:RDEVice:PMETer<Pmtr>:DEFine`

Usage: Event

Manual operation: See "Configured Devices" on page 846

SYSTem:COMMunicate:RDEVice:PMETer:SCAN?

Scans for power meters.

Usage: Query only

Manual operation: See "Scan Instruments" on page 847

SYSTem:COMMunicate:RDEVice:PMETer<Pmtr>:SPCorrection[:STATe]
<Boolean>

Gets/sets the state of the built-in S-parameter correction that is available on R&S NRP-Z power sensors.

Note that this state is persistently stored on the power sensor (and NOT on the R&S ZNB).

See Application Note 1GP70 "Using S-Parameters with R&S®NRP-Z Power Sensors" for background information. This Application Note is available on the Rohde & Schwarz internet site at https://www.rohde-schwarz.com/appnotes/1GP70.

Suffix:
<Pmtr> Number of the configured power meter.

Parameters:
<Boolean> ON | OFF

Manual operation: See "Deembed Two-Port (All Channels)" on page 850

SYSTem:COMMunicate:RDEVice:SMATrix<Matr>:CATalog?

Queries the numbers of the switch matrices currently part of the RF configuration (see `INSTrument:SMATrix` on page 1251). The response is a string containing a comma-separated list of switch matrix numbers.

Suffix:
<Matr> This suffix is ignored and can be omitted.

Example: Suppose switch matrices 1 and 2 are currently connected to the R&S ZNB.
`SYST:COMM:RDEV:SMAT1:CAT?`
Query the numbers of the connected switch matrices. The response is '1, 2'.
`SYST:COMM:RDEV:SMAT1:COUN?`
Query the number of connected switch matrices. The response is 2.

Usage: Query only

SYSTem:COMMunicate:RDEVice:SMATrix:CONFigure:ABORt

Aborts a manual RF connection configuration.

Note: a manual RF connection configuration must be performed within a transaction, enclosed by the commands `SYSTem:COMMunicate:RDEVice:SMATrix:CONFigure:STARt` and `SYSTem:COMMunicate:RDEVice:SMATrix:CONFigure:END`. The modified configuration will not be applied until the end of this transaction.

The new configuration is discarded, i.e. the previous configuration remains active ("rollback"). The transaction wil be terminated.

Usage: Event

SYSTem:COMMunicate:RDEVice:SMATrix:CONFigure:END

Ends a manual RF connection configuration: if the configuration is valid, it will be activated ("commit"). Otherwise an error is returned and the new configuration is discarded, i.e. the previous configuration remains active ("rollback").

In any case the transaction will be terminated.

Note: a manual RF connection configuration must be performed within a transaction, enclosed by the commands `SYSTem:COMMunicate:RDEVice:SMATrix:CONFigure:STARt` and `SYSTem:COMMunicate:RDEVice:SMATrix:CONFigure:END`. The modified configuration will not be applied until the end of this transaction.

The configuration is valid if

- every VNA port is either connected to a matrix port or directly assigned to a DUT test port
- every matrix that is part of the configuration is connected to the VNA by at least one RF connection
- every VNA port that is not connected to a matrix is assigned to a test port
- the test ports are numbered consecutively, starting with 1

Example: See `SYSTem:COMMunicate:RDEVice:SMATrix:CONFigure:STARt`

Usage: Event

SYSTem:COMMunicate:RDEVice:SMATrix:CONFigure:STARt

Starts a manual RF connection configuration.

Note: a manual RF connection configuration must be performed within a transaction, enclosed by the commands `SYSTem:COMMunicate:RDEVice:SMATrix:CONFigure:STARt` and `SYSTem:COMMunicate:RDEVice:SMATrix:CONFigure:END`. The modified configuration will not be applied until the end of this transaction.

An RF connection configuration comprises the RF connections between the VNA, a set of switch matrices and the DUT test ports.

At transaction start, the R&S ZNB creates an in-memory copy of the active configuration and strips off all RF connections from this copy. Subsequent RF connection actions such as `SYSTem:COMMunicate:RDEVice:SMATrix<Matr>:CONFigure:MVNA` and `SYSTem:COMMunicate:RDEVice:SMATrix<Matr>:CONFigure:MTESt` operate on the copy.

However, the copy contains the same switch matrices as the original, so the adequate set of matrices has to be selected beforehand (using manual control or the remote command `INSTrument:SMATrix`).

If a previous transaction is active, it is silently rolled back.

Note that a redefinition of the physical VNA ports (see `[SENSe:]UDSParams<Pt>:PARam`) causes a factory reset and deletes all switch matrix RF connections. So the RF configuration for switch matrices has to be done *after* the port redefinition.

Example: `:INSTrument:SMATrix OFF`
Remove all switch matrices from the RF configuration.
`:SYSTem:COMMunicate:RDEVice:SMATrix:DELete`
Unconfigure all switch matrices, i.e. remove them from the list of configured devices.
`:SYSTem:COMMunicate:RDEVice:SMATrix1:DEFine '', 'ZV-Z82-30', 'LAN', '192.168.0.42'`
Register a single switch matrix of type ZV-Z82-30 (with two 2x5 submatrices).
`:INSTrument:SMATrix ON`
Add the matrix to the RF configuration, performing a default assignment of VNA ports and test ports.
`:SYSTem:COMMunicate:RDEVice:SMATrix:CONFigure:STARt`
Start a manual RF connection configuration.
`:SYSTem:COMMunicate:RDEVice:SMATrix1:CONFigure:MVNA 1,1,3,2,2,3,4,4`
Define non-default matrix-VNA port connections: matrix port 1 to VNA port 1, matrix port 3 to VNA port 2, matrix port 2 to VNA port 3 and matrix port 4 to VNA port 4 (default is `1,3,2,4,3,1,4,2`).
`:SYSTem:COMMunicate:RDEVice:SMATrix1:CONFigure:MTESt 1,1,2,2,9,3,10,4`
Configure 4 matrix test ports only: matrix test port 1 is test port 1, matrix test port 2 is test port 2, matrix test port 9 is test port 3 and matrix test port 10 is test port 4.
`:SYSTem:COMMunicate:RDEVice:SMATrix:CONFigure:END`
Apply the manual RF connection configuration.

Usage: Event

SYSTem:COMMunicate:RDEVice:SMATrix:CONFigure:TVNA <TestPort1>, <VNAPort1>[, <TestPort2>, <VNAPort2>[...]]

Sets/gets the test port connections of the VNA in a switch matrix RF connection setup, i.e. the direct assignments of test ports to physical VNA ports.

Note: a manual RF connection configuration must be performed within a transaction, enclosed by the commands `SYSTem:COMMunicate:RDEVice:SMATrix:CONFigure:STARt` and `SYSTem:COMMunicate:RDEVice:SMATrix:CONFigure:END`. The modified configuration will not be applied until the end of this transaction.

Setting parameters:

<TestPort1>... Number of the test port.
Test ports must be numbered subsequently starting at 1.

<VNAPort1>... Number of the VNA port

Example: `SYSTem:COMMunicate:RDEVice:SMATrix:CONFigure:TVNA 6,2,7,4`
Test Ports 6 and 7 are assigned to VNA ports 2 and 4.

Manual operation: See "Edit Test Port Connection" on page 862

SYSTem:COMMunicate:RDEVice:SMATrix<Matr>:CONFigure:MLVNa {<Matrix VNA Port Label>, <VNA Port Number>}

Sets/gets the RF connections between switch matrix and VNA. Similar to `SYSTem:COMMunicate:RDEVice:SMATrix<Matr>:CONFigure:MVNA`, but uses the front panel labels of the matrix VNA ports instead.

By default, these labels are only available for some matrix types.

Note: a manual RF connection configuration must be performed within a transaction, enclosed by the commands `SYSTem:COMMunicate:RDEVice:SMATrix:CONFigure:STARt` and `SYSTem:COMMunicate:RDEVice:SMATrix:CONFigure:END`. The modified configuration will not be applied until the end of this transaction.

Suffix:

<Matr> Index of the switch matrix (see `SYSTem:COMMunicate:RDEVice:SMATrix<Matr>:DEFine` on page 1559)

Setting parameters:

<Matrix VNA Port Label> These labels are specified in the corresponding matrix driver file (see `SYSTem:COMMunicate:RDEVice:SMATrix<Matr>:DEFine` on page 1559).

<VNA Port Number> Range: 1 ... # of RF ports of the VNA

Example: A switch matrix ZV-Z84-02 is used.
`SYSTem:COMMunicate:RDEVice:SMATrix1:CONFigure:MLVNA '1A',1,'2B',2`
VNA ports 1 and 2 are connected to matrix VNA ports '1A' and '2B', respectively

Manual operation: See "Edit Matrix VNA Port Connections" on page 861

SYSTem:COMMunicate:RDEVice:SMATrix<Matr>:CONFigure:MLTest {<Matrix Test Port Label>, <TPNumber>}

Sets/gets the matrix test port connections. Similar to `SYSTem:COMMunicate:RDEVice:SMATrix<Matr>:CONFigure:MTESt`, but uses the front panel labels of the matrix test ports instead.

By default, these labels are only available for some matrix types.

Note: a manual RF connection configuration must be performed within a transaction, enclosed by the commands `SYSTem:COMMunicate:RDEVice:SMATrix:CONFigure:STARt` and `SYSTem:COMMunicate:RDEVice:SMATrix:CONFigure:END`. The modified configuration will not be applied until the end of this transaction.

Suffix:

<Matr> Index of the switch matrix (see `SYSTem:COMMunicate:RDEVice:SMATrix<Matr>:DEFine` on page 1559)

Setting parameters:

<Matrix Test Port Label> If available, these labels are specified in the corresponding matrix driver file (see `SYSTem:COMMunicate:RDEVice:SMATrix<Matr>:DEFine` on page 1559).

<Test Port Number> Test ports must be numbered subsequently, starting at 1.

Example: A switch matrix ZV-Z82-16 is used.
```
SYSTem:COMMunicate:RDEVice:SMATrix1:CONFigure:MLTESt 'A1',1,'B1',2
```
The first test ports of submatrices A and B are assigned to test ports 1 and 2, respectively.

Manual operation: See "Edit Test Port Connection" on page 862

SYSTem:COMMunicate:RDEVice:SMATrix<Matr>:CONFigure:MTESt
{<MatrixTestPort>, <TestPort>}

Sets/gets the matrix test port connections as a comma-separated list of port numbers

Note: a manual RF connection configuration must be performed within a transaction, enclosed by the commands `SYSTem:COMMunicate:RDEVice:SMATrix:CONFigure:STARt` and `SYSTem:COMMunicate:RDEVice:SMATrix:CONFigure:END`. The modified configuration will not be applied until the end of this transaction.

Suffix:

<Matr> Index of the switch matrix (see `SYSTem:COMMunicate:RDEVice:SMATrix<Matr>:DEFine` on page 1559)

Setting parameters:

{<MatrixTestPort> Number of the switch matrix test port

<TestPort>} Number of the test port.
Test ports must be numbered subsequently starting at 1.

Example: See `SYSTem:COMMunicate:RDEVice:SMATrix:CONFigure:STARt`

Manual operation: See "Edit Test Port Connection" on page 862

SYSTem:COMMunicate:RDEVice:SMATrix<Matr>:CONFigure:MVNA
{<MatrixVNAPort>, <VNAPort>}

Sets/gets the RF connections between switch matrix and VNA as a comma-separated list of port numbers.

Note: a manual RF connection configuration must be performed within a transaction, enclosed by the commands `SYSTem:COMMunicate:RDEVice:SMATrix:CONFigure:STARt` and `SYSTem:COMMunicate:RDEVice:SMATrix:CONFigure:END`. The modified configuration will not be applied until the end of this transaction.

Suffix:
<Matr> Configuration index of the switch matrix (see SYSTem:COMMunicate:RDEVice:SMATrix<Matr>:DEFine on page 1559)

Setting parameters:
{<MatrixVNAPort> Number of the switch matrix VNA port
Range: 1 ... # of VNA ports of the switch matrix

<VNAPort>} Number of the VNA port
Range: 1 ... # of RF ports of the VNA

Example: See SYSTem:COMMunicate:RDEVice:SMATrix:CONFigure:STARt

Manual operation: See "Edit Matrix VNA Port Connections" on page 861

SYSTem:COMMunicate:RDEVice:SMATrix:COUNt?

Gets the number of configured switch matrices (see SYSTem:COMMunicate:RDEVice:SMATrix<Matr>:DEFine on page 1559)

Usage: Query only

Manual operation: See "Configured Devices" on page 856

SYSTem:COMMunicate:RDEVice:SMATrix<Matr>:DEFine <Unused>, <Driver>, <Interface>, <Address>

This command allows

- to configure (i.e. register) a new switching matrix
- to modify the (management) connection settings of an already configured switching matrix
- to get the (management) connection settings of a configured switching matrix

Suffix:
<Matr> Index of the switch matrix.
Switch matrices are numbered in ascending order, starting with 1. If a number is re-used, the current configuration is overwritten.

Parameters:
<Unused> Currently this parameter is ignored and always returns an empty string `''`.

<Driver> Matrix driver name indicating the matrix type (string parameter).
Standard drivers: The supported drivers are displayed in the "Driver" dropdown-list of the "Add External Switchmatrix" dialog. Each list entry corresponds to a matrix driver file (`*.matrix`) located in the `Resources\ExtDev` subdirectory of the analyzer's program directory. Specify the <Driver> as the driver's file name without the `*.matrix` extension, e.g. as `"ZV-Z81-05"`.
Non-standard driver location: You may also specify the full absolute path to an appropriate driver file, e.g. `"C:\Program Files\Rohde-Schwarz\Vector Network Analyzer\ZNB\Resources\ExtDev\ZV-Z81-05.matrix"`.

<Interface> Interface type (string parameter)

'USB'
USB interface

'LAN'
LAN interface

'DIRECTCTRL'
Direct control interface

<Address> Interface-specific address (string parameter).

Example: `:SYST:COMM:RDEV:SMATrix1:DEF '', 'ZN-Z84-42', 'USB', '101142'`
Register switch matrix no.1: type R&S ZN-Z84 (2x24) with serial number `101142` at USB interface.
`:SYST:COMM:RDEV:SMATrix2:DEF '', 'ZV-Z81-05', 'LAN', '10.10.10.10'`
Register switch matrix no.2: type R&S ZV-Z81 (model .05) at IP address 10.10.10.10.
`:SYST:COMM:RDEV:SMATrix3:`
`DEF '', 'ZN-Z84-42', 'DIRECTCTRL', '1319.4500K02::101143'`
Register switch matrix no.3: type R&S ZN-Z84 (2x24) with serial number `101143` at Direct Control interface.

Manual operation: See "Add Device" on page 858

SYSTem:COMMunicate:RDEVice:SMATrix:DELete

Unregisters all switch matrices, i.e. removes them from the list of configured devices.

Example: See `SYSTem:COMMunicate:RDEVice:SMATrix:CONFigure:STARt`

Usage: Event

Manual operation: See "Delete All" on page 858

SYSTem:COMMunicate:RDEVice:SMATrix<Matr>:RELays:SWITch:COUNt?

For mechanical matrices, this command allows to query the list of switch counts for all relays (if supported by the matrix).

Suffix:
<Matr> Index of the switch matrix (see `SYSTem:COMMunicate:RDEVice:SMATrix<Matr>:DEFine` on page 1559)

Usage: Query only

Manual operation: See "Configured External Instruments" on page 830

SYSTem:COMMunicate:RDEVice:SMATrix:SCAN?

Scans for external switch matrices connected via USB or Direct Control.

Returns a comma-separated list `<Driver_1>,<Interface_1>,<Address_1>,...,<Driver_N>,<Interface_N>,<Address_N>`, one triple `<Driver_n>,<Interface_n>,<Address_n>` for each detected switch matrix `<Matr>=1,...,N`.

Use `SYSTem:COMMunicate:RDEVice:SMATrix<Matr>:DEFine` to configure them.

Usage: Query only

Manual operation: See "Scan Instruments" on page 857

SYSTem:COMMunicate:RDEVice:SMATrix<Matr>:SERial?

Returns the serial number of the related switch matrix.

Suffix:
<Matr> Index of the switch matrix (see `SYSTem:COMMunicate:RDEVice:SMATrix<Matr>:DEFine` on page 1559)
If the switch matrix is not part of the RF configuration, the query returns an "Invalid suffix" error.

Usage: Query only

Manual operation: See "Configured Devices" on page 856

7.3.17.2 SYSTem:DATA:MEMory...

These commands allow you to configure memory-mapped trace data transfer. See Section 4.4.3, "Memory-mapped trace data transfer", on page 152.

SYSTem:DATA:MEMory:ADD <TraceName>, <DataFormat>, <NumberOfPoints>

Adds trace <TraceName> to a previously initialized shared memory buffer.

Traces can only be added after the `SYSTem:DATA:MEMory:INITalize` and before the corresponding `SYSTem:DATA:MEMory:COMMit`.

For background information, see Section 4.4.3, "Memory-mapped trace data transfer", on page 152.

Setting parameters:

<TraceName> Trace name. Identifies the trace within the active recall set.

<DataFormat> SDATa | FDATa

The data format of the stored trace values.

SDATa
Unformatted (complex) trace data.

FDATa
Formatted (real-valued) trace data.

<NumberOfPoints> Number of trace values to be stored.

Usage: Setting only

SYSTem:DATA:MEMory:CATalog?

Queries the names of the shared memory buffers initialized using `SYSTem:DATA:MEMory:INITalize`.

Before the `SYSTem:DATA:MEMory:COMMit`, an initialized shared memory buffer has the temporary name `NewMemory`.

For background information, see Section 4.4.3, "Memory-mapped trace data transfer", on page 152.

Usage: Query only

SYSTem:DATA:MEMory:COMMit <ShMemName>

Completes the setup of a named shared memory buffer that was initialized using `SYSTem:DATA:MEMory:INITalize` an configured using one or more `SYSTem:DATA:MEMory:ADD`. The buffer is then available in Windows.

Before the next `COMMit`, the size of the created buffer can be queried using `SYSTem:DATA:MEMory:SIZE?`.

To delete the buffer and free its memory, use `SYSTem:DATA:MEMory:DELete <ShMemName>`, or `SYSTem:DATA:MEMory:RESet`.

For background information, see Section 4.4.3, "Memory-mapped trace data transfer", on page 152.

Setting parameters:

<ShMemName> Gives the shared memory buffer a name, which is required by processes that want to access it.
The name must be unique within the running Windows instance.
The names of the shared memory buffers initialized using `SYSTem:DATA:MEMory:INITalize` can be queried using `SYSTem:DATA:MEMory:CATalog?`.

Usage: Setting only

SYSTem:DATA:MEMory:DELete <ShMemName>

Deletes a shared memory buffer <ShMemName> initialized using `SYSTem:DATA:MEMory:INITalize`.

Before the `SYSTem:DATA:MEMory:COMMit`, an initialized shared memory buffer has the temporary name `NewMemory`. Do not delete it.

Setting parameters:
<ShMemName> Unique name of the shared memory buffer.

Usage: Setting only

SYSTem:DATA:MEMory:INITalize

Initializes a new shared memory buffer.

- Use `SYSTem:DATA:MEMory:ADD` to add traces to it.
- Finally, use `SYSTem:DATA:MEMory:COMMit` to complete the setup and create the named shared memory buffer in Windows.

For background information, see Section 4.4.3, "Memory-mapped trace data transfer", on page 152.

Usage: Event

SYSTem:DATA:MEMory:OFFSet?

Queries the offset in bytes of the last added trace data block, relative to the start of the shared memory buffer (see `SYSTem:DATA:MEMory:ADD`).

The offset is required by processes that want to read the trace data from the buffer.

For background information, see Section 4.4.3, "Memory-mapped trace data transfer", on page 152.

Usage: Query only

SYSTem:DATA:MEMory:RESet

Deletes all shared memory buffers initialized using `SYSTem:DATA:MEMory:INITalize`.

For background information, see Section 4.4.3, "Memory-mapped trace data transfer", on page 152.

Usage: Event

SYSTem:DATA:MEMory:SIZE?

Queries the size of the shared memory buffer that was last created using `SYSTem:DATA:MEMory:COMMit`.

For background information, see Section 4.4.3, "Memory-mapped trace data transfer", on page 152.

Usage: Query only

7.3.17.3 SYSTem:... (other)

The following `SYSTem...` commands provide general instrument configurations.

SYSTem:AGC:GLOBal:METHod....1565
SYSTem:CORRection:JOINcal....1565
SYSTem:CORRection:WIZard[:IMMediate]....1566
SYSTem:DATE....1566
SYSTem:DFPRint?....1567
SYSTem:DISPlay:BAR:HKEY[:STATe]....1567
SYSTem:DISPlay:BAR:MENU[:STATe]....1567
SYSTem:DISPlay:BAR:STATus[:STATe]....1567
SYSTem:DISPlay:BAR:STOols[:STATe]....1567
SYSTem:DISPlay:BAR:TITLe[:STATe]....1567
SYSTem:DISPlay:BAR:TOOLs[:STATe]....1567
SYSTem:DISPlay:COLor....1567
SYSTem:DISPlay:CONDuctances....1568
SYSTem:DISPlay:DIALogs:SETup:MCAL[:STATe]....1568
SYSTem:DISPlay:TRACes:CCOunt....1568
SYSTem:DISPlay:UPDate....1568
SYSTem:DISPlay:XLABels....1569
SYSTem:ERRor:ALL?....1569
SYSTem:ERRor:COUNt?....1569
SYSTem:ERRor:DISPlay:ERRor....1570
SYSTem:ERRor:DISPlay:INFO....1570
SYSTem:ERRor:DISPlay:WARNings....1570
SYSTem:ERRor:DISPlay[:REMote]....1570
SYSTem:ERRor:DISPlay:STATe....1570
SYSTem:ERRor[:NEXT]?....1571
SYSTem:FIRMware:UPDate....1571
SYSTem:FORMat:IDENtify....1571
SYSTem:FPReset....1572
SYSTem:FREQuency?....1572
SYSTem:HELP:HEADers?....1573
SYSTem:HELP:SYNTax?....1573
SYSTem:HELP:SYNTax:ALL?....1573
SYSTem:IDENtify:FACTory....1573
SYSTem:IDENtify[:STRing]....1574
SYSTem:KLOCk....1574
SYSTem:LANGuage....1574
SYSTem:LOGGing:REMote[:STATe]....1575

SYSTem:OPCHeck:PLEVel:PORT<Port>? 1575
SYSTem:OPCHeck:SELFtest? 1576
SYSTem:OPTions:FACTory 1576
SYSTem:OPTions[:STRing] 1576
SYSTem:PASSword[:CENable] 1576
SYSTem:PRESet[:DUMMy] 1577
SYSTem:PRESet:REMote[:STATe] 1577
SYSTem:PRESet:SCOPe 1577
SYSTem:PRESet:STARt 1578
SYSTem:PRESet:USER:CAL 1578
SYSTem:PRESet:USER:NAME 1578
SYSTem:PRESet:USER[:STATe] 1579
SYSTem:REPort:ADD 1579
SYSTem:REPort:SAVE 1580
SYSTem:REPort:STARt 1580
SYSTem:SETTings:UPDate 1580
SYSTem:SHUTdown 1581
SYSTem:SMATrix:OPTimization 1582
SYSTem:SRECorder:CLEar 1582
SYSTem:SRECorder:DATA? 1582
SYSTem:SRECorder[:STATe] 1582
SYSTem:SOUNd:ALARm[:STATe] 1582
SYSTem:SOUNd:STATus[:STATe] 1582
SYSTem:TIME 1583
SYSTem:TRESet[:STATe] 1583
SYSTem:TSLock 1583
SYSTem:TTLout<Pt>:STATus[:STATe] 1584
SYSTem:USER:DISPlay:TITLe 1584
SYSTem:USER:KEY 1585
SYSTem:VERSion? 1585

SYSTem:AGC:GLOBal:METHod <Global AGC Method>

Allows you to select between two AGC algorithms.

Setting parameters:

<Global AGC Method>
POINt | MEMory

POINt
Point AGC

MEMory
Memory AGC

Manual operation: See "Global AGC" on page 821

SYSTem:CORRection:JOINcal <FirstCal>, <LocationFirstCal>, <SecondCal>, <LocationSecondCal>[, <OutputCal>[, <LocationOutputCal>]]

Command for joining calibrations.

- The query returns whether the two calibrations <FirstCal> (from <LocationFirstCal>) and <SecondCal> (from <LocationSecondCal>) are joinable.

- The set command tries to join the two calibrations.
  If successful, it stores the resulting calibration as <OutputCal> in <LocationOutputCal>.

Parameters:

<FirstCal> String parameter with the channel name (for <LocationFirstCal>=CHANnel) or the calibration file name (for <LocationFirstCal>=POOL) of the first calibration.

<LocationFirstCal> CHANnel | POOL

<SecondCal> String parameter with the channel name (for <LocationSecondCal>=CHANnel) or the calibration file name (for <LocationSecondCal>=POOL) of the first calibration.

<LocationSecondCal> CHANnel | POOL

Setting parameters:

<OutputCal> String parameter with the channel name (for <LocationOutputCal>=CHANnel) or the calibration file name (for <LocationOutputCal>=POOL) of the resulting calibration.
Note: If (<OutputCal>,<LocationOutputCal>) matches one of the input calibrations, this input calibration is replaced.

<LocationOutputCal> CHANnel | POOL

Return values:

<Joinable>

Manual operation: See "Input Calibration1/Input Calibration2/Cal Name" on page 660

SYSTem:CORRection:WIZard[:IMMediate] <Dialogs>

Keysight-compatible command to open the Start Cal tab or the Calibration Kits dialog.

Tip: When working with the Section 6.1.3, "GPIB Explorer", on page 870, switch to raw mode ("Options">"Raw mode" in the IECWIN32 GUI) before executing this command.

Setting parameters:

<Dialogs> MAIN | CKIT

MAIN: open the "Calibration > Start Cal" softtool tab
CKIT: open the "Calibration Kits" dialog

Usage: Setting only

SYSTem:DATE <Year>, <Month>, <Day>

The command queries or defines the instrument's current date setting.

The setting command requires administrator rights; refer to Section 3.1.9, "Windows operating system", on page 27.

Parameters:

<Year> Year, four-digit number

<Month> Month, two-digit number, 01 (for January) to 12 (for December)

<Day> Day, two-digit number, 01 to the number of days in the month

Example: `SYST:DATE?`
Response: `2012,05,01` - it is the 1st of May, 2012.

SYSTem:DFPRint?

Queries the device footprint. The device footprint contains detailed information about the instrument and is mostly used for service purposes.

Usage: Query only

Manual operation: See "Save..." on page 826

SYSTem:DISPlay:BAR:HKEY[:STATe] <Boolean>
SYSTem:DISPlay:BAR:MENU[:STATe] <Boolean>
SYSTem:DISPlay:BAR:STATus[:STATe] <Boolean>
SYSTem:DISPlay:BAR:STOols[:STATe] <Boolean>
SYSTem:DISPlay:BAR:TITLe[:STATe] <Boolean>
SYSTem:DISPlay:BAR:TOOLs[:STATe] <Boolean>

Displays or hides the hardkey panel (`HKEY`), the menu bar below the diagram area (`MENU`), the status bar below the diagram area (`STATus`), the softtool panel (`STOols`), the title bar of the main VNA application window (`TITLe`), and the toolbar above the diagram area (`TOOLs`).

Parameters:

<Boolean> ON | OFF

Display or hide the information elements.

Example: `SYSTem:DISPlay:BAR:TOOLs ON; STOols ON; STATus ON`
Display the toolbar, softtool panel, and status bar.
`SYSTem:DISPlay:BAR:TITle OFF; HKEY OFF; MENU OFF`
Hide the title bar, hardkey bar, and menu bar.

Manual operation: See "Tool Bar" on page 808

SYSTem:DISPlay:COLor <ColorScheme>

Selects the color scheme for on-screen display in the active recall set.

Use `HCOPy:PAGE:COLor` to select the print color scheme.

Parameters:

<ColorScheme> UDEFined | DBACkground | LBACkground | BWLStyles | BWSolid

UDEFined
User-defined color scheme

DBACkground
Dark background

LBACkground
Light background

BWLStyles
Black and white line styles

BWSolid
Black and white solid

Example: `SYST:DISP:COL LBAC`
Select the "Light Background" color scheme.

Manual operation: See "Color Scheme" on page 801

SYSTem:DISPlay:CONDuctances <Boolean>

Changes the presentation of "capacitance C<i> in parallel with resistance R<i>" circuit blocks in lumped de/embedding networks.

Parameters:
<Boolean> ON - display conductances
OFF - display capacitaces

Manual operation: See "Conductance in Embedding Networks" on page 816

SYSTem:DISPlay:DIALogs:SETup:MCAL[:STATe] <Boolean>

Enables/disables multiple calibrations in the calibration wizard.

Parameters:
<Boolean>

Manual operation: See "Multiple Cal in Calibration Wizard" on page 814

SYSTem:DISPlay:TRACes:CCOunt <ColorCount>

Defines the maximum number of trace colors (trace properties for the user defined color scheme defined using `DISPlay:CMAP<DispEl>:RGB`.

Parameters:
<ColorCount> Range: 1 to 100

Manual operation: See "Number of Trace Colors" on page 816

SYSTem:DISPlay:UPDate <Activate>

Switches the display on or off while the analyzer is in the remote state. The command has no effect while the analyzer is in the local operating state.

Tip: Switching off the display speeds up the measurement. This command may have an impact on the update of trace and channel settings; see `SYSTem:SETTings:UPDate`.

Parameters:

<Activate> ON | OFF - switch the display on or off. If the display is switched on, the analyzer shows the diagrams and traces like in manual control.
ONCE - switch the display on and show the current trace. This parameter can be used for occasional checks of the measurement results or settings. The measurement is continued, however, the measurement results are not updated. Compared to the `ON` setting, `ONCE` does not slow down the measurement speed.

Example: `SYST:DISP:UPD ON`
Switch the display on to view the traces and diagrams.

SYSTem:DISPlay:XLABels <Boolean>

Enables or disables the display of X-axis grid labels in cartesian diagrams with linear scale.

Parameters:

<Boolean> **ON (1)**
Show X-axis grid labels

OFF (0)
Don't show X-axis grid labels

*RST: OFF

Manual operation: See "Show X-Axis Grid Labels" on page 816

SYSTem:ERRor:ALL?

Queries and at the same time deletes all entries in the error queue.

The entries consist of an error number and a short description of the error. Positive error numbers are instrument-dependent. Negative error numbers are reserved by the SCPI standard; see section Error messages and troubleshooting.

Example: `SYST:ERR:ALL?`
Query all entries in the error queue. `0,"No error"` is returned if the error queue is empty.

Usage: Query only

SYSTem:ERRor:COUNt?

Gets the current number of entries in the SCPI error queue.

Usage: Query only

SYSTem:ERRor:DISPlay:ERRor <Boolean>
SYSTem:ERRor:DISPlay:INFO <Boolean>
SYSTem:ERRor:DISPlay:WARNings <Boolean>

Selectively disables or enables the display of information popups for the related event type (Information, System Error, Warning).

Note that the display of information popups can be globally disabled/enabled using `SYSTem:ERRor:DISPlay:STATe` OFF/ON.

Parameters:

<Boolean> ON | OFF - enable/disable display of information popups for the related event type

*RST: ON

Example: `SYST:ERR:DISP:STATe ON; ERR ON; WARN:OFF; INFO OFF`
Display information popups for system errors, but not for warnings and information messages.

Manual operation: See "Info / Warning / Error" on page 820

SYSTem:ERRor:DISPlay[:REMote] <Boolean>

Disables or enables the display of information popups for remote control errors. These popups appear at the bottom of the remote screen and the manual screen.

Note

- Display of information popups can be globally disabled/enabled using `SYSTem:ERRor:DISPlay:STATe` OFF/ON
- For SCPI error `-113, Undefined header` no tooltip is displayed

Parameters:

<Boolean> ON | OFF - enable/disable display of information popups for remote control errors

*RST: ON

Example: `SYST:ERR:DISP:STATe ON; REM ON`
Switch the display of information popups for remote command errors on.
`FREQ:STAR -1`
Generate a `Remote Error: -222, "Data out of range; ..."` tooltip.

Manual operation: See "Remote Error Info" on page 820

SYSTem:ERRor:DISPlay:STATe <Boolean>

Globally defines whether instrument events shall be indicated by information popups.

Display of popups can be limited to certain event types using commands `SYSTem:ERRor:DISPlay:INFO`, `SYSTem:ERRor:DISPlay:WARNings`, `SYSTem:ERRor:DISPlay:ERRor`, and `SYSTem:ERRor:DISPlay[:REMote]`.

Parameters:

<Boolean> ON | OFF - globally enable or disable the display of information popups
*RST: ON

Example: See `SYSTem:ERRor:DISPlay:ERRor` and `SYSTem:ERRor:DISPlay[:REMote]`.

Manual operation: See "Show Messages" on page 820

SYSTem:ERRor[:NEXT]?

Queries and at the same time deletes the oldest entry in the error queue. Operation is identical to that of `STATus:QUEue[:NEXT]?`

The entry consists of an error number and a short description of the error. Positive error numbers are instrument-dependent. Negative error numbers are reserved by the SCPI standard; see section Error messages and troubleshooting.

Example: `SYST:ERR?`
Query the oldest entry in the error queue. `0,"No error"` is returned if the error queue is empty.

Usage: Query only

SYSTem:FIRMware:UPDate <VNASetupFile>

Installs a firmware version stored in a VNA setup file (`*.exe`) on the analyzer. The installation is automatic and does not require any further action.

Setting parameters:

<VNASetupFile> String variable for the name and directory of a VNA setup file.

Example:
```
SYST:FIRM:UPD 'C:
\Users\Public\Setup\ZNB_ZNBT_3.00.exe'
```
Install firmware version V3.00 from the setup file stored in the public directory of the analyzer's internal hard disk.

Usage: Setting only

SYSTem:FORMat:IDENtify <Option>

Defines the format of the factory ID string.

The available formats present the same information in the same order, but differ slightly w.r.t. separator characters:

- LEGacy format:
  `'Rohde&Schwarz,<VNA Type><Max. Freq>-<Ports>Port, <Order and Serial No>,<FW_Version>`

e.g.
`Rohde-Schwarz,ZNB8-4Port,`
`1311601044100005,3.45.3.67-22.04.2.67`
Order and serial no. are formatted without any separators. The first 10 digits is the order number.

- In NEW format, order and serial no. are separated by a slash, and the order number is presented in "KMAT format", e.g.:
`Rohde-Schwarz,ZNB8-4Port,`
`1311.6010K44/100005,3.45.3.67-22.04.2.67`

Note: To let the instrument identify itself with the factory ID (`*IDN?`), use `SYSTem:IDENtify:FACTory`

Setting parameters:

<Option> LEGacy | NEW

A `*RST` does not change the format.
After a factory reset the LEGacy format is used.

Manual operation: See "Define *IDN + *OPT..." on page 843

SYSTem:FPReset

Performs a factory preset of all instrument settings (i.e. all open recall sets) or of the active recall set, depending on the `SYSTem:PRESet:SCOPe` settings, and deletes channel 1 in the active recall set. As a result, the active recall set contains no channels, traces, and diagram areas.

Example:
`SYST:PRES:SCOP SING`
Define the scope of a preset: the active recall set is reset only.
`SYST:FPR`
Reset the parameters of the current recall set and delete channel 1.

Usage: Event

SYSTem:FREQuency? <MinMax>

Queries the minimum and maximum frequency of the network analyzer. For an overview refer to the tables at the beginning of Section 7.3.14.9, "[SENSe:]FREQuency...", on page 1397.

Tip: In contrast to `[SENSe<Ch>:]FREQuency:STARt?` and the other sweep range commands, `SYSTem:FREQuency?` can be used in all sweep modes.

Query parameters:

<MinMax> MINimum | MAXimum

Return values:

<Frequency> MINimum | MAXimum

Return minimum or maximum frequency.

Example: See `[SENSe<Ch>:]FREQuency:CENTer`

Usage: Query only

Manual operation: See "Start Frequency / Stop Frequency / Center Frequency / Span Frequency" on page 527

SYSTem:HELP:HEADers? [<ParserName>]

Returns a list of all SCPI commands supported by the related parser (or the default parser, if omitted).

Query parameters:
<ParserName> Parser name

Return values:
<Commands> SCPI command headers, one per line

Usage: Query only

SYSTem:HELP:SYNTax? <Command>

This command returns the full syntax of the specified command.

Query parameters:
<Command>

Return values:
<CommandDesc> String containing the command you want to query.

Example: `SYST:HELP:SYNT? 'SYST:ERR?'`
Returns the full syntax. In this case: 'SYSTem:ERRor[:NEXT]'.

Usage: Query only

SYSTem:HELP:SYNTax:ALL?

Queries the implemented SCPI commands and their parameters.

Return values:
<Commands>

Usage: Query only

SYSTem:IDENtify:FACTory

Resets the response to the `*IDN?` query to the factory ID, overwriting a user-defined identification string (`SYSTem:IDENtify`).

Use `SYSTem:FORMat:IDENtify` to change the format of the factory ID string.

Example: `SYSTem:IDENtify:STRing 'MyDevice'; *IDN?`
Define an identity string. The response is `MyDevice`.
`SYSTem:IDENtify:FACTory; *IDN?`
Re-activate the factory setting.

Usage: Event

Manual operation: See "Define *IDN + *OPT..." on page 843

SYSTem:IDENtify[:STRing] <InstId>

Defines an identity string for the network analyzer. The query is equivalent to `*IDN?`.

Parameters:

<InstId> String parameter containing the instrument identity

Example: See `SYSTem:IDENtify:FACTory`

Manual operation: See "Define *IDN + *OPT..." on page 843

SYSTem:KLOCk <Boolean>

Locks or unlocks the local controls of the analyzer. This includes the front panel keys, the keyboard, or other local interfaces.

Parameters:

<Boolean> ON | OFF - lock or unlock the local keys.

*RST: OFF

Example: `SYST:KLOCk ON`
Lock the local keys.

SYSTem:LANGuage <Language>

Defines the remote language of the analyzer, i.e. which commands are available and how they are interpreted.

Also sets:

- The (default) byte order for binary data transfer (see `FORMat:BORDer`)
- The (default) identification (`*IDN?`) and option (`*OPT?`) strings (see `SYSTem:IDENtify:FACTory` and `SYSTem:OPTions:FACTory`).

Parameters:

<Language>

'SCPI'
R&S ZNB-specific command set: the analyzer supports all commands described in this documentation.
Factory default. The byte order is set to `SWAPped`.

'ZVABT' | 'ZVR'
Compatibility mode for network analyzers of the R&S ZVR and R&S ZVA/B/T families.
The byte order is set to `SWAPped`.

'ENA' | 'E5071' | 'PNA' | 'HP8510' | 'HP8530' | 'HP8714' | 'HP8720' | 'HP8753'
Compatibility modes for network analyzers from other manufacturers.
The byte order is set to `NORMal`.

Example: `SYST:LANG 'PNA'`
Select a PNA-compatible command set.

Manual operation: See "Remote Language" on page 843

SYSTem:LOGGing:REMote[:STATe] <Boolean>

Enables logging of all remote control commands transferred to the analyzer.

Parameters:

<Boolean> ON – command sequence stored to file `C:\Users\Public\Documents\Rohde-Schwarz\Vna\RemoteLog\VNARemote.log`.
OFF – command sequence not logged.

Example: `SYST:LOGG:REM ON`
Enable remote logging. The log file `'C:\Users\Public\Documents\Rohde-Schwarz\Vna\RemoteLog\VNARemote.log'` is fixed and does not have to be specified.

SYSTem:OPCHeck:PLEVel:PORT<Port>? <Power>[, <LimitLines>]

Defines and executes a power level operator check on port <Port> and returns its boolean result.

Suffix:

<Port> Physical VNA port no.

Query parameters:

<Power> Source power of port <Port>
Default unit: dBm

<LimitLines> String parameter specifying the name and directory of the limit line file to be loaded.
The default extension (manual control) for limit line files is `*.limit`, although other extensions are allowed. If no path is specified the analyzer searches the current directory, which can be set/queried with `MMEMory:CDIRectory`.
If omitted, the firmware applies default limit lines (±dBm above/below the specified source <Power>.

Return values:

<Result> 1 = PASSED, 0 = FAILED

Usage: Query only

Manual operation: See "Execute/Repeat" on page 837

SYSTem:OPCHeck:SELFtest?

Like the common SCPI command `*TST?`, this operator check performs a self test of the R&S ZNB and returns its result. However, the role of boolean results 0 and 1 is exactly opposite.

Return values:
<Result> 1 = PASSED, 0 = FAILED

Usage: Query only

Manual operation: See "Execute/Repeat" on page 834

SYSTem:OPTions:FACTory

Resets the response to the `*OPT?` query to the factory default value. This command overwrites a user-defined option string; see example.

Example: `SYSTem:OPTions:STRing 'MyOptions'; *OPT?`
Define an identity string. The response is `MyOptions`.
`SYSTem:OPTions:FACTory; *OPT?`
Re-activate the factory setting. The analyzer returns a comma-separated list of software and hardware options.

Usage: Event

Manual operation: See "Define *IDN + *OPT..." on page 843

SYSTem:OPTions[:STRing] <Options>

Defines an option string for the network analyzer. The query is equivalent to `*OPT?`.

Parameters:
<Options> String parameter containing the available options

Example: See `SYSTem:OPTions:FACTory`

Manual operation: See "Define *IDN + *OPT..." on page 843

SYSTem:PASSword[:CENable] <Password>

Sends a password to the analyzer enabling a class of service functions to function (Command ENable). Service functions are activated with the commands of the `DIAGnostic...` system and should be used by a R&S service representative only.

Setting parameters:
<Password> Case-sensitive string variable. Sending an invalid password generates error `-221, (settings conflict)`.

Example: `SYST:PASS "XXXX"`
Enter password.

Usage: Setting only

Manual operation: See "Password" on page 839

SYSTem:PRESet[:DUMMy]

Performs a preset, whose scope is defined using the `SYSTem:PRESet:SCOPe` command. The `SYSTem:PRESet:USER[:STATe]` command determines whether a factory preset or a user-defined preset is performed.

Example: `SYST:PRES:SCOP SING`
Define the scope of a preset: the active recall set is reset only.
`SYST:PRES`
Reset the parameters of the current recall set to their factory defaults.

Manual operation: See "System – [Preset]" on page 866

SYSTem:PRESet:REMote[:STATe] <Boolean>

Defines the behavior of the preset commands `*RST` and `SYSTem:PRESet[:DUMMy]`

Parameters:

<Boolean>

OFF (0)
`*RST` and `SYSTem:PRESet` perform a factory preset.

ON (1)
`*RST` and `SYSTem:PRESet` restore the user preset file selected using `SYSTem:PRESet:USER:NAME`, if this file is available and valid. Otherwise, they perform a factory preset.

Manual operation: See "Align *Rst to Use Defined Preset" on page 813

SYSTem:PRESet:SCOPe <Scope>

Specifies whether the preset actions affect all open recall sets, or only the active recall set.

Applies to the GUI and the `SYSTem:PRESet[:DUMMy]` command. The `*RST` command always resets all open recall sets.

Parameters:

<Scope>

ALL | SINGle

ALL
All open recall sets are deleted and a new recall set"Set1" is created according to the current preset mode.
See `SYSTem:PRESet:USER[:STATe]` on page 1579

SINGle
The active setup is reset according to the current preset mode; the name of the active setup and the parameters of all other setups remain unchanged.

Example: See `SYSTem:PRESet[:DUMMy]`

Manual operation: See "Preset Scope" on page 812

SYSTem:PRESet:STARt <Boolean>

Defines whether the analyzer firmware always starts with the configured system preset (factory or user defined) or with the previous state. This is a global setting.

Parameters:

<Boolean>

ON
The FW always starts with the configured system preset.
See `SYSTem:PRESet:USER:NAME` and `SYSTem:PRESet:USER[:STATe]`.

OFF
The analyzer firmware recalls the previous state, i.e. the settings that were persisted during the previous shutdown.

Manual operation: See "Start in Preset" on page 812

SYSTem:PRESet:USER:CAL <PresetUserCal>

Selects a calibration from the calibration pool that shall be restored during a user-defined preset.

The corresponding cal group file (`<PresetUserCal>.cal`) must be available in the cal pool directory

`C:\Users\Public\Documents\Rohde-Schwarz\Vna\Calibration\Data\`.

Parameters:

<PresetUserCal>

Name of the cal group (i.e. the name of the cal group file without path and extension).
Use the empty string to restore the default behavior (no preset user cal).

Example:

```
MMEMORY:STORE:CORRection 1, 'Refl1 Trans123.cal'
```
Adds the active user calibration to the memory pool
```
SYSTem:PRESet:USER:CAL 'Refl1 Trans123'
```
Sets the newly created cal pool member as the preset user cal.
```
SYSTem:PRESet:USER:CAL?
```
Queries for the preset user cal. Returns `'Refl1 Trans123'`
```
SYSTem:PRESet:USER:CAL ''
```
Restores the default behaviour of the instrument: no preset user cal.

Manual operation: See "Preset User Cal" on page 659

SYSTem:PRESet:USER:NAME <RecallSetFile>

Specifies the name of a recall set file (`*.znx|*.znxml`) to be used for a user-defined preset.

Parameters:

<RecallSetFile> String parameter to specify the absolute or relative path of the recall set file to be loaded. Relative paths are evaluated relative to the current directory (see `MMEMory:CDIRectory`).

Example: See `SYSTem:PRESet:USER[:STATe]`

Manual operation: See "Preset Configuration" on page 812

SYSTem:PRESet:USER[:STATe] <Boolean>

Determines whether a factory or a user-defined preset is performed.

This setting applies to the analyzer GUI and to the remote commands `*RST` and `SYSTem:PRESet[:DUMMy]`, if `SYSTem:PRESet:REMote[:STATe]` is set to `ON`.

Parameters:

<Boolean>

OFF (0)
A factory preset is performed.

ON (1)
A user-defined preset is performed, if the setup file selected using `SYSTem:PRESet:USER:NAME` is available and valid. Otherwise, a factory preset is performed.

Example:

```
SYST:PRES:USER:NAME 'C:\Users\Public\Documents
\Rohde-Schwarz\Vna\RecallSets\Setup_2.znx'
```
Select a setup file for the user-defined preset.
```
SYST:PRES:USER ON
```
Enable the user-defined preset.
```
SYSTem:PRESet:REMote ON
```
Align `*RST` (and `SYSTem:PRESet`) to the user-defined preset.
```
*RST
```
Perform the user-defined preset.

Manual operation: See "Preset Configuration" on page 812

SYSTem:REPort:ADD <Layout>[, <Header>]

Optional second step of the report creation sequence (initiated using `SYSTem:REPort:STARt`).

Adds the diagrams of the active setup to the report, using the specified <Layout> and <Header> text.

Can be repeated multiple times before generating the report using `SYSTem:REPort:SAVE`.

Setting parameters:

<Layout> ALL | SINGle | ACTive | HARDcopy | NONE

ALL
All diagrams on one page

SINGle
One diagram per page

ACT
Active area on one page

HARDcopy
Hardcopy

<Header> Header text of the added diagram pages.

Example: See `SYSTem:REPort:STARt` on page 1580.

Usage: Setting only

Manual operation: See "Add Report" on page 793

SYSTem:REPort:SAVE <PdfFile>

Final step of the report creation sequence.

Generates the report and saves it to a PDF file.

Setting parameters:

<PdfFile> String parameter to specify the name of the PDF file.

Example: See `SYSTem:REPort:STARt` on page 1580.

Usage: Setting only

Manual operation: See "Save Report" on page 793

SYSTem:REPort:STARt

Starts the report creation sequence.

Define the report's contents using `SYSTem:REPort:ADD`. Then save the generated PDF using `SYSTem:REPort:SAVE`.

Example:

```
SYSTem:REPort:STARt
SYSTem:REPort:ADD ACT, 'Unexpeced behavior'
SYSTem:REPort:SAVE 'c:\temp\report.pdf'
```

Usage: Event

Manual operation: See "Start Report" on page 793

SYSTem:SETTings:UPDate <Activate>

Initiates an immediate update of the channel or trace settings.

The command has an effect if the analyzer operates in single sweep mode (INITiate<Ch>:CONTinuous OFF) and if the display update is switched off (SYSTem:DISPlay:UPDate OFF). In this scenario, a change of the channel or trace settings is usually not taken into account immediately. The analyzer waits until the end of the current sweep sequence and changes all settings made during the last sweep period when the next single sweep sequence is initiated. Several settings can be made en bloc, which generally saves time.

SYSTem:SETTings:UPDate ONCE causes the analyzer to apply the settings at once without waiting for the end of the current single sweep sequence. The command has no effect in continuous sweep mode or if the display update is switched on.

The settings are also updated when the continuous sweep mode is activated (INITiate<Ch>:CONTinuous ON).

Setting parameters:

<Activate> ONCE

Causes an immediate update of the settings.

Example: INIT:CONT OFF
Activate single sweep mode.
SYST:SETT:UPD ONCE
Update the settings made during the current single sweep period.

Usage: Setting only

SYSTem:SHUTdown [<Action>]

Performs a shutdown or restart of the FW or OS.

If the optional <Action> parameter is omitted, Windows is shut down after a timeout period of 10 seconds.

After Windows has shut down the instrument is in standby state.

Setting parameters:

<Action> HALT | REBoot | ABORt | CLOSe | RESTart

HALT
Windows is shut down after a timeout period of 20 seconds.

REBoot
Windows is restarted after a timeout period of 20 seconds.

ABORt
Abort a Windows shutdown/restart. This action can only be performed during the timeout period.

CLOSe
Close the firmware.

RESTart
Restart the firmware.

Example: SYST:SHUT
Switch the instrument to standby state.

Usage: Setting only

SYSTem:SMATrix:OPTimization <Unit>

Sets/gets the switch matrix route selection algorithm.

Parameters:

<Unit> SPEed | PRECision

SPEed – Optimizes measurement speed by minimizing the total number of switching procedures. This is highly recommended for matrices with mechanical switches.
PRECision – Optimizes measurement precision by always using best possible routes (according to matrix-specific route prioriza-tion).

SYSTem:SRECorder:CLEar

Clears the SCPI recorder's script editor.

Usage: Event

Manual operation: See "Clear All" on page 907

SYSTem:SRECorder:DATA?

Returns the content of the SCPI recorder's script editor in Block data format.

Usage: Query only

Manual operation: See "Save As Recording" on page 906

SYSTem:SRECorder[:STATe] <Boolean>

Turns SCPI recording ON or OFF.

Parameters:

<Boolean> **OFF (0)**
Not recording

ON (1)
Recording

Manual operation: See "Start Recording Start Recording" on page 905

SYSTem:SOUNd:ALARm[:STATe] <Boolean>
SYSTem:SOUNd:STATus[:STATe] <Boolean>

Deprecated command: Since version 3.45, generating sounds is no longer supported by the analyzer firmware.

These commands switch alarm or status sounds on or off.

Parameters:

<Boolean> ON | OFF

*RST: ON

Example: `SYST:SOUN:ALAR OFF; STAT OFF`
Switch alarm and status sounds off.

SYSTem:TIME <Hours>, <Minutes>, <Seconds>

The command queries or defines the instrument's current time setting.

The setting command requires administrator rights; refer to Section 3.1.9, "Windows operating system", on page 27.

Parameters:

<Hours> Range: 0...23

<Minutes> Range: 0...59

<Seconds> Range: 0...59

Example: `SYST:TIME?`
Response: `12,0,0` - it is precisely 12 pm.

SYSTem:TRESet[:STATe] <Boolean>

Defines the behavior of the `INITiate<Ch>[:IMMediate][:DUMMy]` and `INITiate[:IMMediate]:ALL`.

Parameters:

<Boolean> **OFF (0)**
Previous trace data are preserved.

ON (1)
Previous trace data are deleted.

This is a global setting that is not affected by an instrument reset. Factory default is OFF (0).

Manual operation: See "Restart: Set all Traces to 0" on page 816

SYSTem:TSLock <Type>

Locks the touchscreen functionality of the R&S ZNB, e.g. in order to prevent inadvertent entries during remote control.

Parameters:

<Type> OFF | DIAGrams | SCReen

OFF – touchscreen (e.g. the remote screen) remains active. You can use the buttons in "Remote" softtool panels. If you switch back to manual control, the drag and drop functions (e.g. for markers) are still available.
DIAGrams – lock the drag and drop functions in diagrams .
SCReen – lock the entire screen including the remote screen buttons.

*RST: n/a (*RST does not affect the tochscreen lock setting).

Example: `SYSTem:TSLock SCReen`
Lock the entire screen.

Manual operation: See "Enabled / Lock Diagrams / Lock Screen" on page 809

SYSTem:TTLout<Pt>:STATus[:STATe] <Boolean>

Defines the default values of `CALCulate<Chn>:LIMit:TTLout<Pt>[:STATe]` for new traces.

Suffix:

<Pt> 1 - TTL out pass 1 (pin 13 of USER PORT connector)
2 - TTL out pass 2 (pin 14 of USER PORT connector)

Parameters:

<Boolean> **OFF (0)**
New traces are created with `CALCulate<Chn>:LIMit:TTLout<Pt>[:STATe]` set to `OFF (0)`.

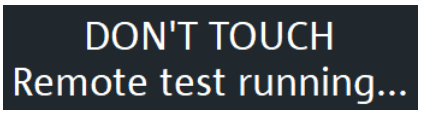
ON (1)
New traces are created with `CALCulate<Chn>:LIMit:TTLout<Pt>[:STATe]` set to `ON (1)`.

*RST: Not affected by *RST; factory default is OFF

Manual operation: See "TTL Pass Default Values" on page 820

SYSTem:USER:DISPlay:TITLe <String>

Defines a title for the remote display, i.e. for the screen that is shown at the instrument, if a remote session is established and the standard display is switched off.



Parameters:

<String> Title string.
A \n in the string starts a new line; see example.

*RST: empty string

Example: `SYST:USER:DISP:TITL "DON'T TOUCH\nRemote test running..."`
Define a title for the remote display.

SYSTem:USER:KEY <Key>[, <Label>]

Labels a user-defined key in the remote display. In the query form the command returns whether or not a user-defined key was tapped or clicked.

Parameters:

<Key> Number of the user key
0 – Delete all user keys and restore the default keys ("Go to Local", "Display Off").
1 to 8 – User key numbers

Range: 0 to 8
*RST: 0

<Label> Label for user key no. 1 to 8 (string variable)

Example: `SYST:USER:KEY 1, 'User S11'`
Define a user key no. 1 labeled S11. The user key is only labeled, no functionality is assigned.
`SYST:USER:KEY? 1`
Query the label. The response is `1, 'User S11'`.
`SYST:USER:KEY?`
Query the user action. The query returns `0,''`, indicating that no user key has been tapped or clicked. If you tap the user softkey no. 1, the response is `1, 'User S11'`. Moreover, the ESR bit no. 6 (User Request) is set.
`SYST:USER:KEY 0`
Delete the user key and restore the default keys.

SYSTem:VERSion?

Returns the SCPI version number to which the analyzer complies. The analyzer complies to the final SCPI version 1999.0

Example: `SYST:VERS?`
Query the SCPI version. The response is `1999.0`.

Usage: Query only

7.3.18 TRACe commands

The TRACe... commands handle active trace data and trace data stored in the analyzer's internal memory.

Trace data formats

Trace data is transferred in either ASCII or block data (REAL) format, depending on the FORMat[:DATA] setting. If the block data format is used, it is recommended to select EOI as receive terminator (SYSTem:COMMunicate:GPIB[:SELF]:RTERminator EOI).

The commands in the TRACe... menu use the following ZVR-compatible parameters to specify traces:

Table 7-19: Reserved Trace Names

Parameter	Meaning	Used in
CH1DATA CH2DATA CH3DATA CH4DATA	Active data trace of channels 1 to 4	TRACe:COPY TRACe[:DATA]:STIMulus[:ALL]? TRACe[:DATA][:RESPonse][:ALL]? CALCulate<Chn>:MATH[:EXPRession][:DEFine]
CH1MEM CH2MEM CH3MEM CH4MEM	Active memory trace associated to the active data trace CH1DATA, CH2DATA, CH3DATA, CH4DATA, respectively.	TRACe[:DATA]:STIMulus[:ALL]? TRACe[:DATA][:RESPonse][:ALL]?
IMPLied	Active data trace, addressed with <Chn>	CALCulate<Chn>:MATH[:EXPRession][:DEFine]
CHMem	Active memory trace assigned to the IMPlied trace	CALCulate<Chn>:MATH[:EXPRession][:DEFine]
MDATA1 MDATA2 MDATA3 MDATA4 MDATA5 MDATA6 MDATA7 MDATA8	Memory trace named Mem<n>[Trc<m>]. The trace name is unique because <n> counts all data and memory traces in the active setup.	TRACe:CLEar TRACe:COPY TRACe[:DATA]:STIMulus[:ALL]? TRACe[:DATA][:RESPonse][:ALL]? CALCulate<Chn>:MATH[:EXPRession][:DEFine]

TRACe:COPY <MemTraceName>, <DataTraceName>

Copies a data trace to a memory trace. The trace to be copied can be specified by two alternative methods:

- As the active data trace of channels 1 to 4 (CH1DATA, CH2DATA, CH3DATA, CH4DATA). If a mathematical trace is active, the associated data trace is copied.

- As a trace with a name (string variable).

The created memory trace can be specified as follows:

- As the memory trace named `Mem<n>[Trc<m>]`, where n = 1, ... 8 and `Trc<m>` is the name of the copied data trace (`MDATA1, MDATA2, MDATA3, MDATA4, MDATA5, MDATA6, MDATA7, MDATA8`).
- As a memory trace with an arbitrary name (string variable).

An existing memory trace with the same name is overwritten.

Note: The copied trace is the data trace which is not modified by any mathematical operations. To copy a mathematical trace to a memory trace, use `TRACe:COPY:MATH`. To copy the active trace to the memory using an automatic memory trace name, use `CALCulate<Chn>:MATH:MEMorize`.

Setting parameters:

<MemTraceName> Name of the memory trace (see also Table 7-19).

Range: <memory_trace> is either a string variable (enclosed in single or double quotes) or one of the following reserved names (no string variables):MDATA1 | MDATA2 | MDATA3 | MDATA4 | MDATA5 | MDATA6 | MDATA7 | MDATA8 (only for memory traces Mem<n>[Trc<m>], where n = 1, ... 8).

<DataTraceName> Name of the data trace (see also Table 7-19).

Range: <data_trace> is either a string variable (enclosed in single or double quotes) or one of the following reserved names (no string variables):CH1DATA | CH2DATA | CH3DATA | CH4DATA (only for the active data trace in channels Ch1, Ch2, Ch3, Ch4).

Example:

```
*RST; :SWE:POIN 20
```

Create a trace with 20 sweep points, making the created trace the active trace of channel 1 (omitted optional mnemonic `SENSe1`).

```
TRAC:COPY "Mem_Pt20",CH1DATA
```

Copy the current state of the created trace to a memory trace named "Mem_Pt20". The memory trace is not displayed.

```
DISP:WIND:TRAC2:FEED "MEM_PT20"
```

Display the created memory trace in the active diagram area (diagram area no. 1).

Usage: Setting only

Manual operation: See "Data to <Destination>" on page 456

TRACe:COPY:MATH <MemTraceName>, <DataTraceName>

Copies a mathematical trace to a memory trace. The trace to be copied can be specified by two alternative methods:

- As the active mathematical trace of channels 1 to 4 (CH1DATA, CH2DATA, CH3DATA, CH4DATA)
- As a trace with a name (string variable).

The created memory trace can be specified as follows:

- As the memory trace named `Mem<n>[Trc<m>]`, where n = 1, ... 8 and `Trc<m>` is the name of the copied data trace (`MDATA1, MDATA2, MDATA3, MDATA4, MDATA5, MDATA6, MDATA7, MDATA8`).
- As a memory trace with an arbitrary name (string variable).

An existing memory trace with the same name is overwritten.

Note: To copy a data trace which is not modified by any mathematical operations, use `TRACe:COPY`

Setting parameters:

<MemTraceName> Name of the memory trace (see also Table 7-19).

Range: <memory_trace> is either a string variable (enclosed in single or double quotes) or one of the following reserved names (no string variables):MDATA1 | MDATA2 | MDATA3 | MDATA4 | MDATA5 | MDATA6 | MDATA7 | MDATA8 (only for memory traces Mem<n>[Trc<m>], where n = 1, ... 8).

<DataTraceName> Name of the data trace (see also Table 7-19).

Range: <data_trace> is either a string variable (enclosed in single or double quotes) or one of the following reserved names (no string variables):CH1DATA | CH2DATA | CH3DATA | CH4DATA (only for the active data trace in channels Ch1, Ch2, Ch3, Ch4).

Example:

```
*RST; :SWE:POIN 20
```

Create a trace with 20 sweep points, making the created trace the active trace of channel 1 (omitted optional mnemonic `SENSe1`).

```
CALC:MATH:SDEF 'Trc1 / 2'; :CALC:MATH:STAT ON
```

Define a mathematical trace, dividing the data trace by 2. Activate the mathematical mode and display the mathematical trace instead of the data trace.

```
TRAC:COPY:MATH 'Mem_Pt20',CH1DATA; :CALC:MATH:STAT OFF
```

Copy the current state of the mathematical trace to a memory trace named "Mem_Pt20". The memory trace is not displayed. Switch the display back to the data trace.

```
DISP:WIND:TRAC2:FEED 'MEM_PT20'
```

Display the created memory trace together with the data trace.

Usage: Setting only

Manual operation: See "Data & Func to <Destination>" on page 457

7.3.19 TRIGger commands

7.3.19.1 TRIGger[:SEQuence]...

The `TRIGger[:SEQuence]...` commands synchronize the analyzer's measurement sequences to external events. These events are indicated to the VNA by sending trigger signals via Trigger In or the USER PORT.

TRIGger<Ch>[:SEQuence]:HOLDoff 1589
TRIGger<Ch>[:SEQuence]:LINK 1589
TRIGger[:SEQuence]:LTRigger[:STATe] 1590
TRIGger[:SEQuence]:LTRigger:COUNt? 1590
TRIGger[:SEQuence]:LTRigger:RESet 1590
TRIGger<Ch>[:SEQuence]:MULTiple:HOLDoff 1591
TRIGger<Ch>[:SEQuence]:MULTiple:SLOPe<Num> 1591
TRIGger<Ch>[:SEQuence]:MULTiple:SOURce 1592
TRIGger<Ch>[:SEQuence]:SLOPe 1592
TRIGger<Ch>[:SEQuence]:SOURce 1593

TRIGger<Ch>[:SEQuence]:HOLDoff <DelayTime>

Sets a delay time between the trigger event and the start of the measurement ("Trigger Delay").

Suffix:
<Ch> Channel number.

Parameters:
<DelayTime> Delay time.

Range: 0 s to 13680 s
Increment: 10 ms
*RST: 0 s
Default unit: s

Example:
```
TRIG:SOUR MAN
```
Activate external trigger source.
```
TRIG:HOLD UP
```
Set a delay time of 10 ms.

Manual operation: See "Delay" on page 555

TRIGger<Ch>[:SEQuence]:LINK <MeasSequence>

Selects the triggered measurement sequence. The identifier for the sequence is a string variable.

Suffix:
<Ch> Channel number

Parameters:
<MeasSequence> Triggered measurement sequence, string variable.
'SWEep' – trigger event starts an entire sweep.
'SEGMent' – trigger event starts a sweep segment, if segmented frequency sweep is active (see example below). If another sweep type is active, the trigger event starts an entire sweep.
'POINt' – trigger event starts measurement at the next sweep point.
'PPOint' – trigger event starts the next partial measurement at the current or at the next sweep point.

*RST: 'SWEep'

Example:
```
SEGM:ADD; :SWE:TYPE SEGM
```
Select segmented frequency sweep.
```
TRIG:LINK 'SEGM'
```
Select a trigger segment as triggered measurement sequence.
```
TRIG:LINK?
```
Query the triggered measurement sequence. The response is `'SEGMENT'`.

Manual operation: See "Sequence" on page 554

TRIGger[:SEQuence]:LTRigger[:STATe] <Boolean>

Activates/deactivates tracking of lost trigger events.

Parameters:
<Boolean> **ON (1)**
Lost trigger traclking active

OFF (0)
Lost trigger tracking inactive

Manual operation: See "Activate Lost Trigger" on page 832

TRIGger[:SEQuence]:LTRigger:COUNt?

Returns the number of lost trigger events the analyzer firmware observed since the last reset of the lost trigger event log (see `TRIGger[:SEQuence]:LTRigger:RESet` on page 1590).

Usage: Query only

Manual operation: See "Reload Lost Trigger Log" on page 832

TRIGger[:SEQuence]:LTRigger:RESet

Clears the lost trigger event log.

Usage: Event

Manual operation: See "Clear Lost Trigger Log" on page 832

TRIGger<Ch>[:SEQuence]:MULTiple:HOLDoff <MeasSequence>[, <DelayTime>]

Sets a delay time between the trigger event and the start of the measurement ("Trigger Delay") in multiple trigger mode.

Suffix:
<Ch> Channel number

Parameters:
<MeasSequence> SWEep | SEGMent | POINt | PPOint

Triggered measurement sequence, `PPOint` denotes "partial measurement"; see `TRIGger<Ch>[:SEQuence]:LINK`.

<DelayTime> Delay time

Range: 0 s to 13680 s
Increment: 10 ms
*RST: 0 s
Default unit: s

Example: See `TRIGger<Ch>[:SEQuence]:MULTiple:SOURce`

Manual operation: See "... |Source| ..." on page 556

TRIGger<Ch>[:SEQuence]:MULTiple:SLOPe<Num> <MeasSequence>[, <Slope>]

Qualifies whether the multiple trigger events occur on the rising or on the falling edge or on the beginning of the high / low level periods of the external TTL trigger signal `<Num>`.

Suffix:
<Ch> Channel number

<Num> Number of trigger signal (1 or 2)

Parameters:
<MeasSequence> SWEep | SEGMent | POINt | PPOint

Triggered measurement sequence, `PPOint` denotes "partial measurement"; see `TRIGger<Ch>[:SEQuence]:LINK`.

<Slope> POSitive | NEGative | HIGH | LOW

Trigger slope for the triggered measurement sequence:
POSitive | NEGative - rising or falling edge
HIGH | LOW - high or low level

Example: See `TRIGger<Ch>[:SEQuence]:MULTiple:SOURce`

Manual operation: See "... |Source| ..." on page 556

TRIGger<Ch>[:SEQuence]:MULTiple:SOURce <Sequence>[, <TrigSource>]

Selects the source of the trigger events the analyzer uses to start a measurement sequence in multiple trigger mode (`TRIGger<Ch>[:SEQuence]:SOURce MULTiple`).

Suffix:
<Ch> Channel number

Parameters:
<Sequence> SWEep | SEGMent | POINt | PPOint

Triggered measurement sequence
`PPOint` denotes "partial measurement"; see `TRIGger<Ch>[:SEQuence]:LINK`.

<TrigSource> IMMediate | EXT1 | EXT2 | E1A2 | E1O2 | MANual

Trigger source for the triggered measurement sequence

IMMediate
Free run measurement (untriggered)

EXT1
Trigger event at Trigger In or Pin 2 of USER PORT

EXT2
Trigger event at pin 25 of USER PORT.

E1A2
Trigger events at EXT1 **AND** EXT2

E1O2
Trigger event at EXT1 **OR** EXT2

MANual
Trigger event generated by pressing the "Manual Trigger" softkey

Example:
```
TRIG:MULT:SOUR SWE, EXT1; SOUR POIN, EXT2
```
Select external trigger 1 as a trigger source for the entire sweep, external trigger 2 as a trigger source for each point.
```
TRIG:MULT:SLOP1 SWE, POS; SLOP2 POIN, POS
```
Trigger on the rising edges of the external trigger signals 1 and 2.
```
TRIG:MULT:HOLD POIN, 1ms
```
Define a trigger delay of 1 ms before each sweep point.

Manual operation: See "... |Source| ..." on page 556

TRIGger<Ch>[:SEQuence]:SLOPe <Slope>

Qualifies whether the trigger event occurs on the rising or on the falling edge or on the beginning of the high / low level periods of the external TTL trigger signal.

Suffix:
<Ch> Channel number

Parameters:

<Slope> POSitive | NEGative | HIGH | LOW

Trigger slope for the triggered measurement sequence:
POSitive | NEGative - rising or falling edge
HIGH | LOW - high or low level

Example: `TRIG:SOUR EXT`
Activate external signal as trigger source.
`TRIG:SLOP NEG`
Trigger on the negative edge of the (external TTL) trigger signal.

Manual operation: See "Signal Type" on page 555

TRIGger<Ch>[:SEQuence]:SOURce <TrigSource>

Selects the source of the trigger events that the analyzer uses to start a measurement sequence.

Suffix:

<Ch> Channel number

Parameters:

<TrigSource> IMMediate | EXTernal | MANual | MULTiple

IMMediate
Free run measurement (untriggered)

EXTernal
Trigger by external signal applied to the Trigger In connector or pin 2 of the USER PORT on the rear panel.

MANual
Trigger event generated by pressing the "Manual Trigger" softkey

MULTiple
Multiple trigger mode, configured by `TRIGger<Ch>[:SEQuence]:MULTiple...` commands

Example: `TRIG:SOUR MAN`
Activate manual trigger mode. The analyzer starts the next measurement sequence when the "Manual Trigger" button is pressed.

Manual operation: See "Free Run (None) / External / Manual / Multiple" on page 553

7.3.19.2 TRIGger:CHANnel:AUXiliary...

The `TRIGger:CHANnel<Ch>:AUXiliary...` commands control the external trigger output Trigger Out at the rear panel of the instrument. The output trigger signal consists of TTL pulses of configurable duration and polarity.

TRIGger:CHANnel<Ch>:AUXiliary:DURation 1594
TRIGger:CHANnel<Ch>:AUXiliary[:ENABle] 1594
TRIGger:CHANnel<Ch>:AUXiliary:INTerval 1595
TRIGger:CHANnel<Ch>:AUXiliary:OPOLarity 1595
TRIGger:CHANnel<Ch>:AUXiliary:POSition 1595

TRIGger:CHANnel<Ch>:AUXiliary:DURation <TrigOutDuration>

Specifies the width of the output trigger pulses at Trigger Out.

The trigger duration must be shorter than the selected sweep interval (see `TRIGger:CHANnel<Ch>:AUXiliary:INTerval`).

If the trigger duration is too long, you can modify one of the following settings:

- Reduce the trigger duration, if possible.
- Slow down the sweep: select a smaller bandwidth or increase the "Sweep Time" setting.

Suffix:
<Ch> Channel number

Parameters:
<TrigOutDuration> Pulse width between 1 µs (1E-6 s) and 1 ms (1E-3 s).
*RST: 1 µs
Default unit: s

Example: See `TRIGger:CHANnel<Ch>:AUXiliary[:ENABle]` on page 1594

Manual operation: See "Duration" on page 558

TRIGger:CHANnel<Ch>:AUXiliary[:ENABle] <EnableTriggOut>

Turns the external trigger output EXT TRIG OUT for channel no. <Ch> on or off.

Suffix:
<Ch> Channel number

Parameters:
<EnableTriggOut> Turn trigger output on (ON) or off (OFF, no signal).
*RST: OFF

Example:
```
TRIGger:CHANnel:AUXiliary:ENABle ON
```
Turn on the trigger output for channel no. 1 (channel suffix omitted).
```
TRIGger:CHANnel:AUXiliary:DURation 10 us
```
Change the width of the output trigger pulses.
```
TRIGger:CHANnel:AUXiliary:POSition BEFore
```
Send the output trigger pulses before data acquisition starts.
```
TRIGger:CHANnel:AUXiliary:INTerval SWEep
```
Send one output trigger pulse per sweep.
```
TRIGger:CHANnel:AUXiliary:OPOLarity POSitive
```
Select positive polarity for the output trigger pulses.

Manual operation: See "Trigger Out Active" on page 557

TRIGger:CHANnel<Ch>:AUXiliary:INTerval <Type>

Specifies whether the trigger output pulses are sent once every sweep (SWEEp), sweep segment (SEGMent), sweep point (POINt), or partial measurement (PPOINt).

Note that POINt changed its meaning from "once every partial measurement" to "once every sweep point" in FW V3.70.

Suffix:
<Ch> Channel number

Parameters:
<Type> SWEep | SEGMent | POINt | PPOint
*RST: SWEep

Manual operation: See "Interval" on page 557

TRIGger:CHANnel<Ch>:AUXiliary:OPOLarity <Type>

Selects the polarity of the output trigger pulses; see Figure 5-29.

Suffix:
<Ch> Channel number

Parameters:
<Type> POSitive | NEGative
*RST: NEGative

Example: See `TRIGger:CHANnel<Ch>:AUXiliary[:ENABle]`

Manual operation: See "Polarity" on page 557

TRIGger:CHANnel<Ch>:AUXiliary:POSition <Type>

Specifies whether the trigger output pulse is sent before or after the selected sweep interval (see `TRIGger:CHANnel<Ch>:AUXiliary:INTerval`).

Suffix:
<Ch> Channel number

Parameters:
<Type> BEFore | AFTer

BEFore
Appropriate if the device needs to be triggered before the data is acquired (typical example: a power meter which is used for power calibration)

AFTer
Appropriate if the device needs to be triggered just after data acquisition (typical example: an external generator which must be reconfigured to get ready for the next measurement)

*RST: AFTer

Example: See TRIGger:CHANnel<Ch>:AUXiliary[:ENABle] on page 1594

Manual operation: See "Position" on page 558

7.4 HUMS and service date commands

This section provides the commands that are related to the Rohde & Schwarz Health and utilization monitoring service (HUMS), which is available on various Rohde & Schwarz instruments.

For background information, see the R&S HUMS User Manual.

- SNMP/REST commands....1596
- System information commands....1600
- Device tags commands....1610
- Utilization commands....1611
- Service date commands....1618

7.4.1 SNMP/REST commands

These commands configure the SNMP and REST interfaces to R&S HUMS.

They all require software option R&S ZNB-980.

SYSTem:COMMunicate:SNMP:COMMunity:RO....1596
SYSTem:COMMunicate:SNMP:COMMunity:RW....1597
SYSTem:COMMunicate:SNMP:CONTact....1597
SYSTem:COMMunicate:SNMP:LOCation....1597
SYSTem:COMMunicate:SNMP:USM:USER....1598
SYSTem:COMMunicate:SNMP:USM:USER:ALL?....1598
SYSTem:COMMunicate:SNMP:USM:USER:DELete....1598
SYSTem:COMMunicate:SNMP:USM:USER:DELete:ALL....1599
SYSTem:COMMunicate:SNMP:VERSion....1599
SYSTem:COMMunicate:REST:ENABle....1599

SYSTem:COMMunicate:SNMP:COMMunity:RO <CommunityString>

Defines the SNMP community string for read-only access.

Prerequisites for this command:

- Select an SNMP version that supports communities (SYSTem:COMMunicate:SNMP:VERSion on page 1599).

Setting parameters:

<CommunityString> String containing the community name.

Example: //Set community name
```
SYST:COMM:SNMP:VERS V12
SYST:COMM:SNMP:COMM:RO 'ABC'
```

Usage: Setting only

Manual operation: See "SNMPv2c Configuration" on page 824

SYSTem:COMMunicate:SNMP:COMMunity:RW <CommunityString>

Defines the SNMP community string for read-write access.

Prerequisites for this command:

- Select an SNMP version that supports communities (`SYSTem:COMMunicate:SNMP:VERSion` on page 1599).

Setting parameters:

<CommunityString> String containing the community name.

Example: //Set read-write access
```
SYST:COMM:SNMP:VERS V12
SYST:COMM:SNMP:COMM:RW 'ABC'
```

Usage: Setting only

Manual operation: See "SNMPv2c Configuration" on page 824

SYSTem:COMMunicate:SNMP:CONTact <SnmpContact>

Defines the SNMP contact information for the administrator.

You can also set the contact information via SNMP if you do not set it via SCPI.

Parameters for setting and query:

<SnmpContact> String containing SNMP contact.

*RST: "" (empty string)

Example: //Set SNMP contact
```
SYST:COMM:SNMP:CONT 'ABC'
```

SYSTem:COMMunicate:SNMP:LOCation <SnmpLocation>

Defines the SNMP location information for the administrator.

You can also set the location information via SNMP if you do not set it via SCPI.

Parameters for setting and query:

<SnmpLocation> String containing SNMP location.

*RST: "" (empty string)

Example: //Return SNMP location
```
SYST:COMM:SNMP:LOC?
```

SYSTem:COMMunicate:SNMP:USM:USER <Name>, <Access>, <Level>[, <Auth_pwd>[, <Priv_pwd>]]

Defines an SNMP user profile.

Prerequisites for this command:

- Select SNMPv3 (`SYSTem:COMMunicate:SNMP:VERSion` on page 1599).

Setting parameters:

<Name> String containing name of the user.

<Access> RO | RW
Defines the access right a user can have.

<Level> NOAuth | AUTH | PRIVacy
Defines the security level.

<Auth_pwd> String containing the authentication password.

<Priv_pwd> String containing the privacy password.

Example:
```
//Create user profile
SYST:COMM:SNMP:VERS V123
SYST:COMM:SNMP:USM:USER 'Peter','RO','PRIV',
'1234','XYZ'
```

Usage: Setting only

Manual operation: See "SNMPv3 Configuration" on page 824

SYSTem:COMMunicate:SNMP:USM:USER:ALL?

Queries the number of users and a list of all SNMP users for SNMPv3.

Prerequisites for this command:

- Select SNMPv3 (`SYSTem:COMMunicate:SNMP:VERSion` on page 1599).

Return values:

<Count> Total number of registered SNMP users.

<Name> List of all user names as a comma-separated list.

Example:
```
//Return all SNMP users
SYST:COMM:SNMP:USM:USER:ALL?
```

Usage: Query only

Manual operation: See "SNMPv3 Configuration" on page 824

SYSTem:COMMunicate:SNMP:USM:USER:DELete <UserName>

Deletes a specific SNMP user profile.

Setting parameters:

<UserName> String containing name of SNMP user profile to be deleted.

Example: //Delete SNMP user profile
```
SYST:COMM:SNMP:USM:USER:DEL "Peter"
```

Usage: Setting only

Manual operation: See "SNMPv3 Configuration" on page 824

SYSTem:COMMunicate:SNMP:USM:USER:DELete:ALL

Deletes all SNMP user profiles.

Example: //Delete all SNMP user profiles
```
SYST:COMM:SNMP:USM:USER:DEL:ALL
```

Usage: Event

Manual operation: See "SNMPv3 Configuration" on page 824

SYSTem:COMMunicate:SNMP:VERSion <SnmpVersion>

Selects the SNMP version.

Parameters for setting and query:

<SnmpVersion> OFF | V12 | V123 | V3 | DEFault

OFF
SNMP communication is off.

V12
SNMP communication with SNMPv2 or lower.

V123
SNMP communication with SNMPv2 and SNMPv3.

V3
SNMP communication with SNMPv3.

*RST: V123

Example: //Select the SNMP version
```
SYST:COMM:SNMP:VERS V12
```

Manual operation: See "SNMP" on page 824

SYSTem:COMMunicate:REST:ENABle <RestState>

Turns communication via the REST API on and off.

Parameters:

<RestState> ON | OFF | 0 | 1

Example: //Return REST state
```
SYST:COMM:REST:ENAB?
```

Manual operation: See "REST" on page 823

7.4.2 System information commands

Using these commands you can retrieve the system information provided by the R&S HUMS.

They all require software option R&S ZNB-980.

DIAGnostic:HUMS[:ALL]?....1600
DIAGnostic:HUMS:BIOS?....1601
DIAGnostic:HUMS:DELete:ALL....1601
DIAGnostic:HUMS:DEVice:HISTory?....1602
DIAGnostic:HUMS:DEVice:HISTory:DELete:ALL....1602
DIAGnostic:HUMS:DEVice:HISTory:EVENt:ADD....1603
DIAGnostic:HUMS:EQUipment?....1603
DIAGnostic:HUMS:FORMat....1604
DIAGnostic:HUMS:SAVE....1604
DIAGnostic:HUMS:SECurity?....1604
DIAGnostic:HUMS:SERVice?....1605
DIAGnostic:HUMS:STATe....1606
DIAGnostic:HUMS:STORage?....1606
DIAGnostic:HUMS:SW?....1607
DIAGnostic:HUMS:SYSTem:INFO?....1608
DIAGnostic:HUMS:SYSTem:STATus?....1609
DIAGnostic:HUMS:SYSTem:STATus:SUMMary?....1609

DIAGnostic:HUMS[:ALL]?

Queries the REST endpoints for the HUMS application in a single query. This allows you to read all HUMS data stored on the instrument via REST API.

The data can be displayed either in `JSON` or `XML` format. For more information about setting the format, see `DIAGnostic:HUMS:FORMat` on page 1604.

Return values:

<Endpoints> <block_data>

String returns REST endpoints as block data in a comma-separated list.
Binary block data with the following syntax:
`#<Digits><Length><Binarydata>`
#
Indicates the start of the binary block
<Digits>
Decimal value
Gives the number of decimal digits used for the `<Length>` value
<Length>
Decimal value
Number of bytes the follow in the `<Binary data>` part
<Binary data>
Binary data in ASCII format

Example: //Return complete HUMS data
`DIAG:HUMS?`

Usage: Query only

DIAGnostic:HUMS:BIOS?

Queries the BIOS information from the instrument. Depending on the set data format, the queried data is either displayed in `XML` or `JSON` format. For more information about setting the data format, see `DIAGnostic:HUMS:FORMat` on page 1604.

Return values:

<BIOSInfo> <block_data>

BIOS information of the connected instrument as block data in a comma-separated list.
`#<manufacturer>,<serialNumber>,<version>,<releaseDate>,<caption>,<isPresent>`
Binary block data with the following syntax:
`#<Digits><Length><Binarydata>`
#
Indicates the start of the binary block
<Digits>
Decimal value
Gives the number of decimal digits used for the `<Length>` value
<Length>
Decimal value
Number of bytes the follow in the `<Binary data>` part
<Binary data>
Binary data in ASCII format

Example: //Return BIOS information
`DIAG:HUMS:BIOS?`

Usage: Query only

DIAGnostic:HUMS:DELete:ALL

Deletes the complete HUMS data. This includes device history, device tags, SCPI connections, utilization history and utilizations.

Example: //Delete HUMS data
`DIAG:HUMS:DEL:ALL`

Usage: Event

Manual operation: See "Delete HUMS History" on page 824

DIAGnostic:HUMS:DEVice:HISTory?

Queries the device history information of the connected instrument. Depending on the set data format, the queried data is either displayed in `XML` or `JSON` format. For more information about setting the data format, see `DIAGnostic:HUMS:FORMat` on page 1604.

Return values:

<HistoryInfo> <block_data>

Device history information of the connected instrument as block data in a comma-separated list:
`#blockdata[{event1},{event2},{event3}...]`
With the following parameters:
`<eventID>,<eventTimestamp>,<eventMessage>, <eventDetails>,<eventSeverity>`
Binary block data with the following syntax:
`#<Digits><Length><Binarydata>`
#
Indicates the start of the binary block
<Digits>
Decimal value
Gives the number of decimal digits used for the `<Length>` value
<Length>
Decimal value
Number of bytes the follow in the `<Binary data>` part
<Binary data>
Binary data in ASCII format

Example: //Return device history
`DIAG:HUMS:DEV:HIST?`
Returns for example:

```
#44715[{"eventId":32,"eventTimestamp":
"2021-02-02T17:25:39Z","eventMessage":
"Deviation from Self Alignment Temperature",
"eventDetails":
"Deviations resolved","eventSeverity":0}
```

Usage: Query only

DIAGnostic:HUMS:DEVice:HISTory:DELete:ALL

Deletes the complete device history information of the connected instrument.

Example: //Delete complete device history
`DIAG:HUMS:DEV:HIST:DEL:ALL`

Usage: Event

DIAGnostic:HUMS:DEVice:HISTory:EVENt:ADD <Severity>, <Message>[, <Detail>]

Adds an event to the event history.

Note that the event ID, time stamp and event source are automatically created when you add the event.

Setting parameters:

<Severity> INFO | WARNing | ERRor
Severity of the event.

<Message> Short description of the event.

<Detail> More comprehensive description of the event.

Example:
```
//Add an event ot the event history
DIAG:HUMS:DEV:HIST:EVEN:ADD
INFO,"InfoEvent","An event that has occurred on
the instrument"
```

Usage: Setting only

DIAGnostic:HUMS:EQUipment?

Queries the equipment information (device footprint) of the instrument. Depending on the set data format, the queried data is either displayed in `XML` or `JSON` format. For more information about setting the data format, see `DIAGnostic:HUMS:FORMat` on page 1604.

Return values:

<EquipmentInfo> <block_data>
Device footprint of the connected instrument as block data in a comma-separated list.
Binary block data with the following syntax:
`#<Digits><Length><Binarydata>`
#
Indicates the start of the binary block
<Digits>
Decimal value
Gives the number of decimal digits used for the `<Length>` value
<Length>
Decimal value
Number of bytes the follow in the `<Binary data>` part
<Binary data>
Binary data in ASCII format

Example:
```
//Return device footprint
DIAG:HUMS:EQU?
```

Usage: Query only

DIAGnostic:HUMS:FORMat <DataFormat>

Selects the format for the queried HUMS data. You can query the HUMS data either in JSON format or XML format.

The defined format affects all other commands that return block data.

Parameters:

<DataFormat> JSON | XML

JSON
Returns the HUMS data in JSON format.

XML
Returns the HUMS data in XML format.

*RST: JSON

Example: //Return data in JSON format
```
DIAG:HUMS:FORM JSON
```

DIAGnostic:HUMS:SAVE <path>

Saves the HUMS history as a ZIP file to your preferred path.

Setting parameters:

<path>

Example: //Save HUMS history data
```
DIAG:HUMS:SAVE 'C:\HUMS\hums_2021.zip'
```

Usage: Setting only

Manual operation: See "Export HUMS History" on page 824

DIAGnostic:HUMS:SECurity?

Queries the security information of the instrument. Depending on the set data format, the queried data is either displayed in XML or JSON format. For more information about setting the data format, see DIAGnostic:HUMS:FORMat on page 1604.

Return values:

<SecurityInfo> <block_data>

Security information of the connected instrument as block data in a comma-separated list.
```
#blockdata{"antimalware":
{<name>,<enabled>,<upToDate>,<timestamp>},
"firewallEnabled"}
```
Binary block data with the following syntax:
```
#<Digits><Length><Binarydata>
```
#
Indicates the start of the binary block
<Digits>
Decimal value

Gives the number of decimal digits used for the <Length> value
<Length>
Decimal value
Number of bytes the follow in the <Binary data> part
<Binary data>
Binary data in ASCII format

Example:
```
//Return security information
DIAG:HUMS:SEC?
#3133{"antimalware":{"name":
"Windows Defender","enabled":false,"upToDate":
true,"timestamp":
"2018-02-08T10:09:22Z"},"firewallEnabled":true}
```

Usage: Query only

DIAGnostic:HUMS:SERVice?

Queries the service information of the instrument. Depending on the set data format, the queried data is either displayed in XML or JSON format. For more information about setting the data format, see DIAGnostic:HUMS:FORMat on page 1604.

Return values:

<ServiceInfo> <block_data>

Service information of the connected instrument as block data in a comma-separated list:
#blockdata{<lastdate>,<requiredservice>,<calibration>:
{<lastcalibration>,<nextDue>,<nextDueExpired>,
<recommendedCalibrationInterval>},<uptime>}
Binary block data with the following syntax:
#<Digits><Length><Binarydata>
#
Indicates the start of the binary block
<Digits>
Decimal value
Gives the number of decimal digits used for the <Length> value
<Length>
Decimal value
Number of bytes the follow in the <Binary data> part
<Binary data>
Binary data in ASCII format

Example: //Return service information
```
DIAG:HUMS:SERV?
```
Returns for example:
```
#3196{"last":"2021-01-19T23:00:00Z","required":
2,"calibration":{"last":
"2021-01-19T23:00:00Z","nextDue":
"2022-01-19T23:00:00Z","nextDueExpired":
false,"recommendedCalibrationInterval":
"P1Y"},"uptime":0}
```

Usage: Query only

DIAGnostic:HUMS:STATe <State>

Turns the HUMS service and data collection on and off.

Parameters:

<State> ON | OFF | 1 | 0

*RST: ON

Example: //Turn on HUMS service
```
DIAG:HUMS:STAT ON
```

Manual operation: See "State (Device Utilization)/Status" on page 823

DIAGnostic:HUMS:STORage?

Queries the storage information of the instrument. Depending on the set data format, the queried data is either displayed in `XML` or `JSON` format. For more information about setting the data format, see `DIAGnostic:HUMS:FORMat` on page 1604.

Return values:

<StorageInfo> <block_data>

Returns storage information for each storage device of the connected instrument as block data.
```
#blockdata[{storagedevice1},
{storagedevice2},...]
```
Binary block data with the following syntax:
```
#<Digits><Length><Binarydata>
```
#
Indicates the start of the binary block
<Digits>
Decimal value
Gives the number of decimal digits used for the `<Length>` value
<Length>
Decimal value
Number of bytes the follow in the `<Binary data>` part
<Binary data>
Binary data in ASCII format
For each storage device it returns:

<diskname>,<temperature>,<type>,<modelfamily>,
<model>,<serialnumber>,<id>,<firmwareversion>,
<capacity>,<sectorsize>,<rotationrate>,<formfactor>,
<ataversion>,<sataversion>,<smartdatatable>.
For each value of the smart data table it returns:
<id>,<attributename>,<dataflag>,<value>,
<worst>,<threshold>,<type>,<update>,
<whenfailed>,<rawvalue>

Example: //Return storage information
DIAG:HUMS:STOR?
Returns for example:
#44620[{"name":"/dev/sda","temperature":
305,"type":"ata","modelFamily":null,"model":
"SanDisk SD9SB8W128G","serialNumber":
"000000000000","id":
"5 001b44 8b67d0a7d","firmwareVersion":
"A1234000","userCapacity":
128035676160,"sectorSizeLogical":
512,"sectorSizePhysical":512,"rotationRate":
"0","formFactor":"2.5 inches","ataVersion":
"ACS-4 T13/BSR INCITS 529 revision 5","sataVersion":
"SATA 3.3","smartDataTable":[{"id":
5,"attributeName":
"Reallocated_Sector_Ct","dataFlag":50,"value":
100,"worst":100,"threshold":0,"type":
"Old_age","updated":
"Always","whenFailed":null,"rawValue":0},{"id":
9,"attributeName":"Power_On_Hours","dataFlag":
50,"value":100,"worst":100,"threshold":
0,"type":"Old_age","updated":
"Always","whenFailed":null,"rawValue":16856}]

Usage: Query only

DIAGnostic:HUMS:SW?

Queries information about the installed software on the instrument. Depending on the set data format, the queried data is either displayed in XML or JSON format. For more information about setting the data format, see DIAGnostic:HUMS:FORMat on page 1604.

Return values:

<SoftwareInfo> <block_data>

For each installed software package it returns the following information as block data in a comma-separated list:
#blockdata[{<index>,<name>,<softwareType>,
<installDate>}]
Binary block data with the following syntax:
#<Digits><Length><Binarydata>
#

Indicates the start of the binary block
<Digits>
Decimal value
Gives the number of decimal digits used for the <Length> value
<Length>
Decimal value
Number of bytes the follow in the <Binary data> part
<Binary data>
Binary data in ASCII format

Example: //Return installed software information
DIAG:HUMS:SW?

Usage: Query only

DIAGnostic:HUMS:SYSTem:INFO?

Queries the system information of the instrument. Depending on the set data format, the queried data is either displayed in XML or JSON format. For more information about setting the data format, see DIAGnostic:HUMS:FORMat on page 1604.

Return values:

<SystemInfo> <block_data>

Returns system information of the connected instrument as block data.
Binary block data with the following syntax:
#<Digits><Length><Binarydata>
#
Indicates the start of the binary block
<Digits>
Decimal value
Gives the number of decimal digits used for the <Length> value
<Length>
Decimal value
Number of bytes the follow in the <Binary data> part
<Binary data>
Binary data in ASCII format

Example: //Return system information
DIAG:HUMS:SYST:INFO?

Usage: Query only

DIAGnostic:HUMS:SYSTem:STATus?

Queries the complete system status information of the instrument. Depending on the set data format, the queried data is either displayed in `XML` or `JSON` format. For more information about setting the data format, see `DIAGnostic:HUMS:FORMat` on page 1604.

Return values:

<SystemStatus> <block_data>

Returns complete system status information of the connected instrument as block data in a comma-separated list:
`#blockdata{<globalStatus>,<tablevalues>:[{tablevalue1},{tablevalue2},...]}`
Binary block data with the following syntax:
`#<Digits><Length><Binarydata>`
#
Indicates the start of the binary block
<Digits>
Decimal value
Gives the number of decimal digits used for the `<Length>` value
<Length>
Decimal value
Number of bytes the follow in the `<Binary data>` part
<Binary data>
Binary data in ASCII format
For each table value it returns the following values:
`<id>,<description>,<descriptionExtended>,<value>,<unit>,<upperLimit>,<lowerLimit>,<reference>,<severity>`

Example:

```
//Return system status information
DIAG:HUMS:SYST:STAT?
#41874{"globalStatus":3,"values":[{"id":
31522816,"description":
"RF Overload","descriptionExtended":
null,"value":null,"unit":null,"upperLimit":
null,"lowerLimit":null,"reference":
null,"severity":1},...}]}
```

Usage: Query only

DIAGnostic:HUMS:SYSTem:STATus:SUMMary?

Queries the status summary of the complete system.

Return values:

<StatusSummary> **OK**
The system is OK.

WARNing
There are some issues with the system which might be critical later.

ERRor
Problems with the system occurred and the status is critical.

Example: //Return status summery of the system
`DIAG:HUMS:SYST:STAT:SUMM?`

Usage: Query only

7.4.3 Device tags commands

DIAGnostic:HUMS:TAGS:ALL? 1610
DIAGnostic:HUMS:TAGS[:VALue] 1610
DIAGnostic:HUMS:TAGS:DELete 1611
DIAGnostic:HUMS:TAGS:DELete:ALL 1611

DIAGnostic:HUMS:TAGS:ALL?

Queries all key-value tags that you have assigend to the instrument. Depending on the set data format, the queried data is either displayed in `XML` or `JSON` format. For more information about setting the data format, see `DIAGnostic:HUMS:FORMat` on page 1604.

Return values:
<ID> ID number of the defined tag.

<Key> String containing key name of the defined tag.

<Value> String containing value of the defined tag.

Example: //Return all tags
```
DIAG:HUMS:TAGS:ALL?
1,"location","building_11",2,"time zone","CET"
```

Usage: Query only

Manual operation: See "Add / Delete (Trash) / Delete All" on page 825

DIAGnostic:HUMS:TAGS[:VALue] <ID>, <Key>, <Value>
DIAGnostic:HUMS:TAGS[:VALue]? <ID>

Adds or modifies a key-value pair (device tag).

The query returns the key-value pair for a given ID or an empty string if the ID is unknown.

Parameters:
<Key> String containing key name of the queried tag.

<Value> String containing value of the queried tag.

Parameters for setting and query:

<ID> 0 - 31
ID number of the tag you want to modify or query.
To identify the ID number, query all device tags from the system first. For more information, read here `DIAGnostic:HUMS:TAGS:ALL?` on page 1610.

Example: //Add or modify a tag (tag 1)
`DIAG:HUMS:TAGS 1,'location','building_11'`

Manual operation: See "Add / Delete (Trash) / Delete All" on page 825

DIAGnostic:HUMS:TAGS:DELete <ID>

Deletes a certain tag you assigned to your instrument, including its key and value.

Setting parameters:

<ID> ID number of the tag you want to delete.
To identify the ID number, query all device tags from the system first. For more information, see `DIAGnostic:HUMS:TAGS:ALL?` on page 1610.

Example: //Delete tag
`DIAG:HUMS:TAGS:DEL 0`

Usage: Setting only

Manual operation: See "Add / Delete (Trash) / Delete All" on page 825

DIAGnostic:HUMS:TAGS:DELete:ALL

Deletes all key-value tags you have assigned to the instrument.

Example: //Delete all tags
`DIAG:HUMS:TAGS:DEL:ALL`

Usage: Event

Manual operation: See "Add / Delete (Trash) / Delete All" on page 825

7.4.4 Utilization commands

Using these commands you can retrieve the utilization information provided by the R&S HUMS.

They all require software option R&S ZNB-980.

DIAGnostic:HUMS:UTILization? 1612
DIAGnostic:HUMS:UTILization:ACTivity:TRACking:STATe 1613
DIAGnostic:HUMS:UTILization:CUSTom:ADD 1613
DIAGnostic:HUMS:UTILization:CUSTom:ALL? 1614
DIAGnostic:HUMS:UTILization:CUSTom:DELete 1614
DIAGnostic:HUMS:UTILization:CUSTom:DELete:ALL 1615

DIAGnostic:HUMS:UTILization:CUSTom:UPDate 1615
DIAGnostic:HUMS:UTILization:HISTory? 1615
DIAGnostic:HUMS:UTILization:HISTory:DELete:ALL 1617
DIAGnostic:HUMS:UTILization:HISTory:DETailed? 1617

DIAGnostic:HUMS:UTILization?

Queries the current utilization data of the instrument.

Depending on the set data format, the queried utilization data is either displayed in `XML` or `JSON` format. For more information about setting the data format, see `DIAGnostic:HUMS:FORMat` on page 1604.

Return values:

<UtilizationData> <block_data>

Returns the current utilization data of the connected instrument as block data.
`#blockdata[{utilization1},{utilization2},...]`
Binary block data with the following syntax:
`#<Digits><Length><Binarydata>`
#
Indicates the start of the binary block
<Digits>
Decimal value
Gives the number of decimal digits used for the `<Length>` value
<Length>
Decimal value
Number of bytes the follow in the `<Binary data>` part
<Binary data>
Binary data in ASCII format
For each utilization it returns the following values:
`<id>,<scope>,<reference>,<unit>,<description>,<value>,<startvalue>,<state>`

Example:

```
//Return utilization data
DIAG:HUMS:UTIL?
Returns for example:
#123456[{"id":31719936,"scope":
"DEVICE_SPECIFIC","reference":null,"name":
"Self alignments","unit":
"counter","description":
"Report number of self alignments","value":
0,"startupValue":
0,"activityTracking":true},...]
```

Usage: Query only

DIAGnostic:HUMS:UTILization:ACTivity:TRACking:STATe <ID>, <State>
DIAGnostic:HUMS:UTILization:ACTivity:TRACking:STATe? <ID>

Turns the activity tracking of utilizations on and off.

Parameters:

<State> ON | OFF | 1 | 0

ON | 1
Tracking activity = on

OFF | 0
Tracking activity = off

Parameters for setting and query:

<ID> ID number of the utilization activity.
To find out the ID of the utilization, you have to query the complete utilization data. For more information, see DIAGnostic:HUMS:UTILization? on page 1612.

Example: //Set activity tracking state of ID 31719936

```
DIAG:HUMS:UTIL:ACT:TRAC:STAT 31719936,'1'
```

DIAGnostic:HUMS:UTILization:CUSTom:ADD <ID>, <Name>, <Description>, <Unit>, <State>

Creates a custom utilization.

Setting parameters:

<ID> Identifier of the custom utilization.
Range: 1 to 99

<Name> Name of the custom utilization that appears in the user interface.

<Description> Description of the custom utilization.

<Unit> Unit of the custom utilization that appears in the user interface.

<State> ON | OFF | 1 | 0
Initial activity state of custom utilization.

Example:
```
//Create a custom utilization
DIAG:HUMS:UTIL:CUST:ADD
1,"TrackingSomething","This utilization tracks
something","Hz",1
//Set an initial absolute value for the utilization with ID = 1; initial
start value = 0
DIAG:HUMS:UTIL:CUST:UPD 1,25,abs
//Upon an event, the utilization changes its value each time the
event occurs
DIAG:HUMS:UTIL:CUST:UPD 1,5,incr
//New value = 30, new start value = 25
//Another event
DIAG:HUMS:UTIL:CUST:UPD 1,5,incr
//New value = 35, new start value = 30 etc.
//Query all available custom utilizations
DIAG:HUMS:UTIL:CUST:ALL
//Delete the custom utilization with ID = 1
DIAG:HUMS:UTIL:CUST:DEL 1
//Delete all custom utilizations
DIAG:HUMS:UTIL:CUST:DEL:ALL
```

Usage: Setting only

DIAGnostic:HUMS:UTILization:CUSTom:ALL?

Queries all custom utilizations currently in use.

Return values:

<block_data> String returns REST endpoints as block data in a comma-separated list.
Binary block data with the following syntax:
`#<Digits><Length><Binarydata>`
#
Indicates the start of the binary block
<Digits>
Decimal value
Gives the number of decimal digits used for the `<Length>` value
<Length>
Decimal value
Number of bytes the follow in the `<Binary data>` part
<Binary data>
Binary data in ASCII format

Example: See `DIAGnostic:HUMS:UTILization:CUSTom:ADD`

Usage: Query only

DIAGnostic:HUMS:UTILization:CUSTom:DELete <ID>

Deletes a single custom utilization.

Setting parameters:

<ID> ID of the utilization you want to delete.

Example: See DIAGnostic:HUMS:UTILization:CUSTom:ADD

Usage: Setting only

DIAGnostic:HUMS:UTILization:CUSTom:DELete:ALL

Deletes all custom utilizations.

Example: See DIAGnostic:HUMS:UTILization:CUSTom:ADD

Usage: Event

DIAGnostic:HUMS:UTILization:CUSTom:UPDate <ID>, <Value>, <Mode>[, <State>]

Changes the current values of the custom utilization (and indirectly also the start value).

Setting parameters:

<ID> Identifier of the custom utilization.

<Value> Defines the value to be changed.

<Mode> **ABSolute**
Defines an absolute value for the utilization.
For example: Currently, the utilization value = 5. If you define an absolute value of 1, the new value is 1.

INCRement
Increases or decreases the current value by a certain amount.
For example: Currently, the utilization value = 5. If you increment it by 1, the new value is 6.
If you you specify a negative number, it decreases the current value.

In both cases, the previous value becomes the new start value.

<State> ON | OFF | 1 | 0

Example: See DIAGnostic:HUMS:UTILization:CUSTom:ADD

Usage: Setting only

DIAGnostic:HUMS:UTILization:HISTory? [<StartDate>[, <EndDate>[, <TimeResolution>]]]

Queries the history of utilization activity on the connected instrument.

The data format is either XML or JSON, depending on DIAGnostic:HUMS:FORMat.

Timestamps follow the Unix time format.

Query parameters:

<StartDate> String containing the start date of the evaluation period (ISO8601 format).
If you do not define a start date, the command returns data starting with the data acquisition one month before the end date.

<EndDate> String containing the end date of the evaluation period (ISO8601 format).
If you do not define a end date, the command returns data for a month starting on the start date.

<Granularity> Resolution of the evaluation period (default = usage over a period of 1 day or 86 400 seconds).
You can define the resolution directly in terms of seconds as a number or indirectly in terms of SEC, MIN, HOUR or DAY (for example `7DAY` results in a resolution of one week). The base unit is seconds.

Default unit: s

Return values:

<History> <block_data>

Utilization usage of the connected instrument during the time period you have defined. The number of return values depends on the selected time period and time resolution. For example, if you evaluate over a time period of one week with a resolution of one day, the command returns 7 values.
Binary block data with the following syntax:
`#<Digits><Length><Binarydata>`
#
Indicates the start of the binary block.
<Digits>
Decimal value
Gives the number of decimal digits used for the `<Length>` value.
<Length>
Decimal value
Number of bytes that follow in the `<Binary data>` part.
<Binary data>
Binary data in ASCII format
The binary data represents the utilization activity. Each utilization is assigned a timestamp for when the utilization has been active. Together they form a pair of values.
`#blockdata{"timestamps":`
`[<timestamp1>,<timestamp2>,...],"activity":`
`[<activity1>,<activity2>,...]}`
Each timestamp therefore has a corresponding activity:
`<timestamp1> + <activity1>, <timestamp2> + <activity2>`

Example: //Query utilization history for all utilizations
```
DIAG:HUMS:UTIL:HIST?
```
Returns for example:
```
#3440{"timestamps":
[1612259226,1612345626,1612432026,1612518426],"activity":
[0,0,66000,81000]}
```

Usage: Query only

DIAGnostic:HUMS:UTILization:HISTory:DELete:ALL

Deletes the complete utilization history information from the instrument.

Alternatively, you can delete:

- device history only with `DIAGnostic:HUMS:DEVice:HISTory:DELete:ALL`
- all data with `DIAGnostic:HUMS:DELete:ALL`

Example: //Delete complete utilization history
```
DIAG:HUMS:UTIL:HIST:DEL:ALL
```

Usage: Event

DIAGnostic:HUMS:UTILization:HISTory:DETailed? <ID>[, <StartDate>[, <EndDate>[, <TimeResolution>]]]

Query parameters:

<ID> ID of the utilization you want to get information about.

<StartDate> String containing the start date of the evaluation period (ISO8601 format).
If you do not define a start date, the command returns data starting with the data acquisition one month before the end date.

<EndDate> String containing the end date of the evaluation period (ISO8601 format).
If you do not define a end date, the command returns data for a month starting on the start date.

<Granularity> Resolution of the evaluation period (default = usage over a period of 1 day or 86 400 seconds).
You can define the resolution directly in terms of seconds as a number or indirectly in terms of SEC, MIN, HOUR or DAY (for example `7DAY` results in a resolution of one week). The base unit is seconds.

Default unit: s

Return values:

<History> <block_data>

Utilization usage of the connected instrument during the time period you have defined. The number of return values depends on the selected time period and time resolution. For example, if you evaluate over a time period of one week with a resolution of one day, the command returns 7 values.
Binary block data with the following syntax:
`#<Digits><Length><Binarydata>`
#
Indicates the start of the binary block.
<Digits>
Decimal value
Gives the number of decimal digits used for the `<Length>` value.
<Length>
Decimal value
Number of bytes that follow in the `<Binary data>` part.
<Binary data>
Binary data in ASCII format
The binary data represents the utilization activity. Each utilization is assigned a timestamp for when the utilization has been active. Together they form a pair of values.
`#blockdata{"timestamps":`
`[<timestamp1>,<timestamp2>,...],"activity":`
`[<activity1>,<activity2>,...]}`
Each timestamp therefore has a corresponding activity:
`<timestamp1> + <activity1>, <timestamp2> + <activity2>`

Example: //Return utilization history for a specific utilization
`DIAG:HUMS:UTIL:HIST? 1`
Returns for example:
`#3440{"timestamps":`
`[1612259226,1612345626,1612432026,1612518426],"activity":`
`[0,0,6000,18000]}`

Usage: Query only

7.4.5 Service date commands

Using these commands you can get and set instrument service dates.

These dates are part of the R&S HUMS information, but do not require software option R&S ZNB-980.

DIAGnostic:SERVice:CALibration:DATE 1619
DIAGnostic:SERVice:CALibration:DUE:DATE 1619
DIAGnostic:SERVice:CALibration:DUE:STATe? 1619
DIAGnostic:SERVice:CALibration:INTerval? 1619
DIAGnostic:SERVice:DATE 1620
DIAGnostic:SERVice:STATe? 1620

DIAGnostic:SERVice:CALibration:DATE <CalibrationDate>

Defines last date and time the instrument was calibrated in ISO 8601 format.

Parameters:
<CalibrationDate> String containing calibration date of the instrument.

Example: //Set calibration date
`DIAG:SERV:CAL:DATE "2019-05-05T00:00:00Z"`

Manual operation: See "Last Calibration" on page 833

DIAGnostic:SERVice:CALibration:DUE:DATE <DueDate>

Defines next date and time the instrument needs calibration to be done in ISO 8601 format. The response may be empty in case of no fixed next calibration due.

Parameters:
<DueDate> String containing next calibration due date.
An empty string resets the date (= no due date).

Example: //Set calibration due date
`DIAG:SERV:CAL:DUE:DATE "2020-05-12T00:00:00Z"`

Manual operation: See "Next Calibration Due" on page 832

DIAGnostic:SERVice:CALibration:DUE:STATe?

Queries the state if the calibration is OK or the instrument requires calibration.

Return values:
<State> **NAN**
Not a number, e.g. if no due date is set.
OK
Calibration is OK since due date has not expired yet.
EXPired
Calibration due date has expired. Calibration is needed.

Example: //Return calibration due state
`DIAG:SERV:CAL:DUE:STAT?`

Usage: Query only

DIAGnostic:SERVice:CALibration:INTerval? <Duration>

Queries the recommended calibration interval, set by Rohde & Schwarz service.

Parameters:
<Duration> String indicating the calibration interval (ISO 8601 duration)

Example: `DIAG:SERV:CAL:INT?`
returns `"P1Y"` if the recommended calibration interval is 1 year

Usage: Query only

Manual operation: See "Recommended Calibration Interval" on page 832

DIAGnostic:SERVice:DATE <ServiceDate>

Defines the last date and time the instrument was serviced (ISO 8601 format).

Parameters:

<ServiceDate> String containing last service date.

Example: //Return last service date
```
DIAG:SERV:DATE?
```

Manual operation: See "Last Service" on page 832

DIAGnostic:SERVice:STATe?

Queries the state if the instrument requires service.

Return values:

<ServiceState>

NAN
Not a number, e.g. if no service date is set.

OK
Instrument is OK. No service is needed.

DEViations
Deviations have been detected. Service might be required soon.

REQuired
Instrument requires service.

Example: //Return service state
```
DIAG:SERV:STAT?
```

Usage: Query only

7.5 R&S ZVR/ZVABT compatible commands

The commands in this chapter are supported for compatibility with analyzers of the R&S ZVR and R&S ZVAB family; they do not introduce any new functionality. For new programs, it is recommended to use the commands in Section 7.3, "SCPI command reference", on page 917.

CALCulate<Chn>:LIMit:CONTrol:DOMain....1621
CALCulate<Chn>:LIMit:RDOMain:COMPlex....1622
CALCulate<Chn>:LIMit:RDOMain:FORMat....1623
CALCulate<Chn>:LIMit:RDOMain:SPACing....1623
CALCulate<Chn>:LIMit:LOWer:STATe....1623
CALCulate<Chn>:LIMit:UPPer:STATe....1623
CALCulate<Ch>:PARameter:DEFine....1624
CALCulate<Chn>:MARKer<Mk>:FUNCtion:BWIDth....1625
CALCulate<Chn>:MARKer<Mk>:FUNCtion:DELTa:STATe....1626

CALCulate<Chn>:MARKer<Mk>:FUNCtion[:SELect] 1626
CALCulate<Chn>:MARKer<Mk>:REFerence:FUNCtion[:SELect] 1626
CALCulate<Chn>:MARKer<Mk>:FUNCtion:TARGet 1627
CALCulate<Chn>:MARKer<Mk>:REFerence:FUNCtion:TARGet 1627
CALCulate<Chn>:MARKer<Mk>:MAXimum 1627
CALCulate<Chn>:MARKer<Mk>:MINimum 1627
CALCulate<Chn>:MARKer<Mk>:REFerence:MAXimum 1627
CALCulate<Chn>:MARKer<Mk>:REFerence:MINimum 1627
CALCulate<Chn>:MARKer<Mk>:SEARch:LEFT 1628
CALCulate<Chn>:MARKer<Mk>:SEARch:NEXT 1628
CALCulate<Chn>:MARKer<Mk>:SEARch:RIGHt 1628
CALCulate<Chn>:MARKer<Mk>:REFerence:SEARch:LEFT 1628
CALCulate<Chn>:MARKer<Mk>:REFerence:SEARch:NEXT 1628
CALCulate<Chn>:MARKer<Mk>:REFerence:SEARch:RIGHt 1628
CALCulate<Chn>:MARKer<Mk>:SEARch[:IMMediate] 1628
CALCulate<Chn>:MARKer<Mk>:REFerence:SEARch[:IMMediate] 1628
CALCulate<Chn>:MATH[:EXPRession][:DEFine] 1629
DIAGnostic:SERVice:FUNCtion 1629
FORMat:DEXPort:SOURce 1630
INPut<PhyPt>:ATTenuation 1630
INSTrument[:SELect] 1631
OUTPut<Chn>:DPORt 1631
[SENSe<Ch>:]CORRection:CKIT:<ConnType>:<StandardType> 1632
[SENSe:]CORRection:CKIT:INSTall 1633
[SENSe<Ch>:]CORRection:COLLect[:ACQuire] 1633
[SENSe<Ch>:]CORRection:COLLect:METHod 1634
[SENSe<Ch>:]CORRection:COLLect:SAVE:DEFault 1635
[SENSe<Ch>:]CORRection:COLLect:SAVE[:DUMMy] 1636
[SENSe<Ch>:]CORRection:DATA 1637
[SENSe<Ch>:]CORRection:OFFSet<PhyPt>:MAGNitude 1639
[SENSe<Ch>:]FREQuency:CONVersion:MIXer:IFFixed 1639
[SENSe<Ch>:]FREQuency:CONVersion:MIXer:LOFixed 1640
[SENSe<Ch>:]FREQuency:CONVersion:MIXer:RFFixed 1640
[SENSe<Ch>:]FREQuency:MODE 1641
[SENSe<Ch>:]SEGMent<Seg>:CLEar 1641
[SENSe<Ch>:]SEGMent<Seg>:OVERlap 1641
[SENSe<Ch>:]SWEep:SPACing 1642
[SENSe<Chn>:]FUNCtion[:ON] 1642
SOURce<Ch>:POWer<PhyPt>:CORRection:COLLect[:ACQuire] 1644
TRACe:CLEar 1644
TRACe[:DATA][:RESPonse][:ALL]? 1645
TRACe[:DATA]:STIMulus[:ALL]? 1645

CALCulate<Chn>:LIMit:CONTrol:DOMain <SweepType>

Deletes the existing limit line and (re-)defines the physical units of the stimulus values of the limit line. The units of the response values and the scaling of the y-axis can be defined via `CALCulate<Ch>:LIMit:RDOMain:...`

Suffix:
<Chn> Channel number used to identify the active trace.

Setting parameters:
<SweepType> FLIN | FLOG | FSEG | FSINgle | TLIN | TLOG | PLIN | PLOG | PSINgle

Keywords for the units of the stimulus values. The selected unit must be compatible with the sweep type ([SENSe<Ch>:]SWEep:TYPE); otherwise the limit line can not be displayed and no limit check is possible.
The parameters form three groups: FLIN, FLOG, FSEG, and FSINgle select frequency units for the limit line. TLIN and TLOG select time units, PLIN, PLOG and PSINgle select power units.

*RST: FLIN
Default unit: Hz (for FLIN, FLOG, FSEG, FSINGle); s (for TLIN, TLOG), dBm (for PLIN, PLOG, PSINgle).

Example:
```
SWE:TYPE POW
```
Select a power sweep.
```
CALC:LIM:CONT:DOM PLIN
```
Delete all existing limit line segments and select level units for the limit line of the active trace.
```
CALC:LIM:CONT -20, -10
```
Define a limit line segment in the stimulus range between -20 dBm and -10 dBm.

Usage: Setting only

CALCulate<Chn>:LIMit:RDOMain:COMPlex <UnitRef>

Deletes the existing limit line and (re-)defines the physical units of the response values of the limit line. The units of the stimulus values are defined via CALCulate<Chn>:LIMit:CONTrol:DOMain.

Tip: This command is complemented by CALCulate<Chn>:LIMit:RDOMain:FORMat and CALCulate<Chn>:LIMit:RDOMain:SPACing.

Suffix:
<Chn> Channel number used to identify the active trace.

Setting parameters:
<UnitRef> S | SINV | Y | Z | YREL | ZREL

Keyword for the physical unit of the response values. The parameters form four groups: S and SINV select relative units (dB) for the limit line. Y selects admittance units (S/Siemens). Z selects impedance units (Ω). YREL and ZREL select dimensionless numbers (U).

Usage: Setting only

CALCulate<Chn>:LIMit:RDOMain:FORMat <UnitRef>

Deletes the existing limit line and (re-)defines the physical units of the response values of the limit line. The units of the stimulus values are defined via `CALCulate<Chn>:LIMit:CONTrol:DOMain`.

Tip: This command is complemented by `CALCulate<Chn>:LIMit:RDOMain:COMPlex` and `CALCulate<Chn>:LIMit:RDOMain:SPACing`.

Suffix:
<Chn> Channel number used to identify the active trace.

Setting parameters:
<UnitRef> COMPlex | MAGNitude | PHASe | REAL | IMAGinary | SWR | GDELay | L | C

Keyword for the physical unit of the response values. The parameters form four groups: COMPlex, REAL, IMAGinary, and SWR select dimensionless numbers (U) for the limit line. MAGNitude selects relative units (dB). PHASe selects phase units (deg). GDELay selects time units (s). L selects inductance units (H/Henry). C selects capacitance units (F/Farad).

Usage: Setting only

CALCulate<Chn>:LIMit:RDOMain:SPACing <Format>

Deletes the existing limit line and (re-)defines the physical units of the response values of the limit line. The units of the stimulus values are defined via `CALCulate<Chn>:LIMit:CONTrol:DOMain`.

Tip: This command is complemented by `CALCulate<Chn>:LIMit:RDOMain:COMPlex` and `CALCulate<Chn>:LIMit:RDOMain:FORMat`.

Suffix:
<Chn> Channel number used to identify the active trace.

Setting parameters:
<Format> LINear | LOGarithmic | DB | SIC

Keyword for the physical unit of the response values.

Default unit: dB (irrespective of the parameter selected)

Usage: Setting only

CALCulate<Chn>:LIMit:LOWer:STATe <Boolean>
CALCulate<Chn>:LIMit:UPPer:STATe <Boolean>

These commands switch the lower and upper limit check on or off. Lower limit line segments are assigned even numbers; upper limit line segments are assigned odd numbers; see `CALCulate<Chn>:LIMit:LOWer[:DATA]` and `CALCulate<Chn>:LIMit:UPPer[:DATA]`. `CALCulate<Chn>:LIMit:LOWer:STATe` does not affect segments with odd numbers; `CALCulate<Chn>:LIMit:UPPer:STATe` does not affect segments with even numbers.

Note: Use `CALCulate<Chn>:LIMit:STATe` to switch on or off the entire limit check, including upper and lower limit lines.

Suffix:
<Chn> Channel number used to identify the active trace.

Parameters:
<Boolean> ON | OFF - switch limit check on or off.
*RST: OFF

Example:
```
CALC:LIM:LOW -10, 0, 0, -10
```
Define two limit line segments covering the entire sweep range. Two upper limit line segments with default response values are created in addition.
```
CALC:LIM:UPP 0, 5, 5, 0
```
Change the response values of the upper limit line segments.
```
CALC:LIM:LOW:STAT ON; :CALC:LIM:UPP:STAT ON; :
CALC:LIM:FAIL?
```
Switch the limit check on and query the result.

Manual operation: See "Limit Check" on page 485

CALCulate<Ch>:PARameter:DEFine <TraceName>, <Result>[, <TestPortNum>]

Creates a trace and assigns a channel number, a name and a measurement parameter to it. The trace is not displayed. To display a trace defined via `CALCulate<Ch>:PARameter:DEFine`, a window must be created (`DISPlay[:WINDow<Wnd>][:STATe] ON`) and the trace must be assigned to this window (`DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:FEED`); see example below.

Traces must be selected to become active traces; see `CALCulate<Ch>:PARameter:SELect`.

Note: The parameter names in this command differ from R&S ZNB conventions; moreover the parameter list is not complete. The alternative command `CALCulate<Ch>:PARameter:SDEFine` uses a complete parameter list with compatible names.

Suffix:
<Ch> Channel number. `<Ch>` may be used to reference a previously defined channel. If <Ch> does not exist, it is generated with default channel settings.

Setting parameters:
<TraceName> Trace name, e.g. 'Trc4'. See "Rules for trace names" in "Table Area" on page 453.

<Result> S11 | S12 | S13 | S14 | S21 | S22 | S23 | S24 | S31 | S32 | S33 | S34 | S41 | S42 | S43 | S44 | A | B | C | D | R1 | R2 | R3 | R4 | AB | AC | AD | BA | BC | BD | CA | CB | CD | DA | DB | DC | AR1 | AR2 | AR3 | AR4 | BR1 | BR2 | BR3 | BR4 | CR1 | CR2 | CR3 | CR4 | DR1 | DR2 | DR3 | DR4 | R1A | R1B | R1C | R1D | R2A | R2B | R2C | R2D | R3A | R3B | R3C | R3D | R4A | R4B | R4C | R4D | R1R2 | R1R3 | R1R4 | R2R1 | R2R3 | R2R4 | R3R1 | R3R2 | R3R4 | R4R1 | R4R2 | R4R3

Measurement parameter; see list of parameters below.

<TestPortNum> Test port number, drive port for wave quantities and ratios, ignored for S-parameters.

Example: `CALC4:PAR:DEF 'Ch4Tr1', S11`
Create channel 4 and a trace named Ch4Tr1 to measure the input reflection coefficient S11.
`DISP:WIND:STAT ON`
Create diagram area no. 1.
`DISP:WIND:TRAC2:FEED 'CH4TR1'`
Display the generated trace in diagram area no. 1, assigning a trace number 2.

Usage: Setting only

The measurement parameter is selected by means of the following keywords (the selection depends on the number of test ports of the analyzer, e.g. `S44` is not available on 2-port analyzers):

S11 | S12 | S13 | S14 | S21 | S22 | S23 | S24 | S31 | S32 | S33 | S34 | S41 | S42 | S43 | S44	S-parameters
A | B | C | D	Wave quantities b_1, b_2, b_3, b_4 (received waves)
R1 | R2 | R3 | R4	Wave quantities a_1, a_2, a_3, a_4 (reference waves)
AB | AC | AD | BA | BC | BD | CA | CB | CD | DA | dB | DC	Ratio of wave quantities b_1/b_2, b_1/b_3, ... , b_4/b_3 (received waves only)
AR1 | AR2 | AR3 | AR4 | BR1 | BR2 | BR3 | BR4 | CR1 | CR2 | CR3 | CR4 | DR1 | DR2 | DR3 | DR4 | R1A | R1B | R1C | R1D | R2A | R2B | R2C | R2D | R3A | R3B | R3C | R3D | R4A | R4B | R4C | R4D	Ratio of wave quantities b_1/a_1, b_1/a_2, ... , b_4/a_4, a_1/b_1, a_1/b_2, ..., A_4/B_4 (received waves to reference waves or reference waves to received waves)
R1R2 | R1R3 | R1R4 | R2R1 | R2R3 | R2R4 | R3R1 | R3R2 | R3R4 | R4R1 | R4R2 | R4R3	Ratio of wave quantities a_1/a_2, a_1/a_3, ... , a_4/a_3 (reference waves only)

CALCulate<Chn>:MARKer<Mk>:FUNCtion:BWIDth <Bandwidth>

Defines the bandfilter level, i.e. the minimum excursion for the bandpass and bandstop peaks.

Tip: Use `CALCulate<Chn>:MARKer<Mk>:BWIDth` to set the bandwidth and query the results of a bandfilter search. Note the sign convention for input values.

Suffix:
<Chn> Channel number used to identify the active trace.

<Mk> This numeric suffix is ignored and may be set to any value because the bandfilter search functions always use markers M1 to M4.

Parameters:

<Bandwidth>
Range: -100 dB to 100 dB
Increment: 0.03 dB
*RST: 3 dB
Default unit: dB

Example: See `CALCulate<Chn>:MARKer<Mk>:BWIDth`

CALCulate<Chn>:MARKer<Mk>:FUNCtion:DELTa:STATe <Boolean>

Switches the delta mode for marker `<Mk>` on trace no. `<Chn>` on or off.

Note: This command is the ZVR-compatible equivalent of `CALCulate<Chn>:MARKer<Mk>:DELTa[:STATe]`.

Suffix:

<Chn> Channel number used to identify the active trace.

<Mk> Marker number.

Parameters:

<Boolean>
ON | OFF - enable or disable the delta mode.
*RST: OFF

CALCulate<Chn>:MARKer<Mk>:FUNCtion[:SELect] <Mode>
CALCulate<Chn>:MARKer<Mk>:REFerence:FUNCtion[:SELect] <Mode>

Selects a search mode for the related marker.

The search can then be initiated using one of the commands:

- `CALCulate<Ch>:MARKer<Mk>:SEARch...`
- `CALCulate<Ch>:MARKer:REFerence:SEARch...`
- `CALCulate<Ch>:MARKer<Mk>:MAXimum` or `CALCulate<Ch>:MARKer<Mk>:MINimum`
- `CALCulate<Ch>:MARKer:REFerence:MAXimum` or `CALCulate<Ch>:MARKer:REFerence:MINimum`

The marker must be created before using `CALCulate<Chn>:MARKer<Mk>[:STATe] ON` or `CALCulate<Chn>:MARKer<Mk>:REFerence[:STATe] ON`.

Note: This command is not needed except for compatibility with ZVR programs. Use `CALCulate<Chn>:MARKer<Mk>:FUNCtion:EXECute` or `CALCulate<Chn>:MARKer<Mk>:REFerence:FUNCtion:EXECute` to select a search mode and at the same time initiate the search.

Suffix:

<Chn> Channel number used to identify the active trace.

<Mk> Marker number.
Ignored for reference markers and for bandfilter searches (`BFILter`), because bandfilter searches always use markers M1 to M4.

Parameters:

<Mode> NONE | MAXimum | MINimum | RPEak | LPEak | NPEak | TARGet | LTARget | RTARget | BFILter | MMAXimum | MMINimum | SPRogress

See `CALCulate<Chn>:MARKer<Mk>:FUNCtion:EXECute`

CALCulate<Chn>:MARKer<Mk>:FUNCtion:TARGet <SearchValue>
CALCulate<Chn>:MARKer<Mk>:REFerence:FUNCtion:TARGet <SearchValue>

Defines the target value for a target search with the related marker (see "Marker addressing" on page 1032), which can be activated using `CALCulate<Chn>:MARKer<Mk>:FUNCtion:EXECute TARGet` or `CALCulate<Chn>:MARKer<Mk>:REFerence:FUNCtion:EXECute TARGet`.

Note: These commands are the ZVR-compatible equivalents of `CALCulate<Chn>:MARKer<Mk>:TARGet` and `CALCulate<Chn>:MARKer<Mk>:REFerence:TARGet`.

Suffix:

<Chn> Channel number used to identify the active trace.

<Mk> Marker number (ignored for reference markers)

Parameters:

<SearchValue> Target search value of marker no. `<Mk>`.

Range: Depending on the format of the active trace. For a dB Mag trace the range is -200 dB to +200 dB.
Increment: 0.1 dB
*RST: Depending on the trace format; 0 dB for a dB Mag trace.
Default unit: dB

CALCulate<Chn>:MARKer<Mk>:MAXimum
CALCulate<Chn>:MARKer<Mk>:MINimum
CALCulate<Chn>:MARKer<Mk>:REFerence:MAXimum
CALCulate<Chn>:MARKer<Mk>:REFerence:MINimum

These commands select a search mode for the related marker (see "Marker addressing" on page 1032) and initiate a maximum or minimum search.

The marker must be created before using `CALCulate<Chn>:MARKer<Mk>[:STATe] ON` or `CALCulate<Chn>:MARKer<Mk>:REFerence[:STATe] ON`.

Note: These commands are the ZVR-compatible equivalents of `CALCulate<Chn>:MARKer<Mk>:FUNCtion:EXECute MINimum | MAXimum` and `CALCulate<Chn>:MARKer<Mk>:REFerence:FUNCtion:EXECute MINimum | MAXimum`.

Suffix:
<Chn> Channel number used to identify the active trace.

<Mk> Marker number (ignored for reference markers)

Usage: Event

CALCulate<Chn>:MARKer<Mk>:SEARch:LEFT
CALCulate<Chn>:MARKer<Mk>:SEARch:NEXT
CALCulate<Chn>:MARKer<Mk>:SEARch:RIGHt
CALCulate<Chn>:MARKer<Mk>:REFerence:SEARch:LEFT
CALCulate<Chn>:MARKer<Mk>:REFerence:SEARch:NEXT
CALCulate<Chn>:MARKer<Mk>:REFerence:SEARch:RIGHt

These commands selects a search mode for the related marker (see "Marker addressing" on page 1032), and initiate a search for the next valid peak to the left, the next highest or lowest value among the valid peaks, and the next valid peak to the right.

The marker must be created before using `CALCulate<Chn>:MARKer<Mk>[:STATe] ON` or `CALCulate<Chn>:MARKer<Mk>:REFerence[:STATe] ON`.

Note: These commands are the ZVR-compatible equivalents of `CALCulate<Chn>:MARKer<Mk>:FUNCtion:EXECute LPEak | NPEak | RPEak` and `CALCulate<Chn>:MARKer<Mk>:REFerence:FUNCtion:EXECute LPEak | NPEak | RPEak`.

Suffix:
<Chn> Channel number used to identify the active trace.

<Mk> Marker number (ignored for reference markers)

Usage: Event

CALCulate<Chn>:MARKer<Mk>:SEARch[:IMMediate]
CALCulate<Chn>:MARKer<Mk>:REFerence:SEARch[:IMMediate]

Initiates a search for the related marker (see "Marker addressing" on page 1032).

Before using this command:

- The marker must be created using `CALCulate<Chn>:MARKer<Mk>[:STATe] ON` or `CALCulate<Chn>:MARKer<Mk>:REFerence[:STATe] ON`.
- The search function must be selected using `CALCulate<Chn>:MARKer<Mk>:FUNCtion[:SELect]` or `CALCulate<Chn>:MARKer<Mk>:REFerence:FUNCtion[:SELect]`.

Note: Together with `CALCulate<Chn>:MARKer<Mk>:FUNCtion[:SELect]` and `CALCulate<Chn>:MARKer:REFerence:FUNCtion[:SELect]`, these commands are the ZVR-compatible equivalents of `CALCulate<Chn>:MARKer<Mk>:FUNCtion:EXECute` and `CALCulate<Chn>:MARKer<Mk>:REFerence:FUNCtion:EXECute`.

Suffix:
<Chn> Channel number used to identify the active trace.

<Mk> Marker number
Ignored not only for reference markers, but also for bandfilter and multiple peak searches, where a whole marker set is calculated for each sweep.

Usage: Event

CALCulate<Chn>:MATH[:EXPRession][:DEFine] <Expression>

Defines a simple mathematical relation between traces. To calculate and display the new mathematical trace, the mathematical mode must be switched on (`CALCulate<Chn>:MARKer<Mk>[:STATe] ON`).

Note: This command places some restrictions on the mathematical expression and the operands. Use `CALCulate<Chn>:MATH[:EXPRession]:SDEFine` to define general expressions.

Suffix:
<Chn> Channel number used to identify the active trace.

Parameters:
<Expression> <expr>
(`<operand1><operator1><operand2>[<operator2><operand3>]`) The expression must be enclosed in brackets.
Operands: See list of trace names in Section 7.3.18, "TRACe commands", on page 1586.
Operators: +, -, *, /

Example: `*RST; CALC:MATH:MEM`
Copy the current state of the default trace Trc1 to a memory trace named "Mem2[Trc1]". The memory trace is not displayed.
`CALC:MATH (CH1DATA / MDATA2)`
Define a mathematical trace, dividing the data trace by the stored memory trace. The mathematical trace is not displayed
`CALC:MATH:STAT ON`
Display the mathematical trace instead of the active data trace.

DIAGnostic:SERVice:FUNCtion <SFId1>, <SFId2>...

Activates a service function (mainly for internal use). Service functions are identified by groups of numbers, separated by dots.

Parameters:
<SFId1>

<SFId2> Service function identifier entered as a (pseudo-)numeric value, the dots being replaced by commas. Five groups of numbers are allowed at maximum. See also `DIAGnostic:SERVice:SFUNction`.

FORMat:DEXPort:SOURce <Format>

Defines the format for traces retrieved with the R&S ZVR-compatible command `TRACe[:DATA][:RESPonse][:ALL]?`.

This command is not relevant for results read with the `CALCulate<Chn>:DATA...` commands.

Parameters:

<Format> FDATa | SDATa | MDATa

See list of parameters below. The unit is the default unit of the measured parameter; see `CALCulate<Ch>:PARameter:SDEFine`.

Range: Depending on the measured parameter and format.
*RST: SDATa

Example: See `TRACe[:DATA][:RESPonse][:ALL]?`

The following parameters are related to trace data:

FDATa	Formatted trace data, according to the selected trace format (`CALCulate<Chn>:FORMat`). 1 value per trace point for Cartesian diagrams, 2 values for polar diagrams.
SDATa	Unformatted trace data: Real and imaginary part of each measurement point. 2 values per trace point irrespective of the selected trace format. The trace mathematics is not taken into account.
MDATa	Unformatted trace data (see `SDATa`) after evaluation of the trace mathematics.

INPut<PhyPt>:ATTenuation <Attenuation>

Sets the attenuation for the received waves. This command is available if at least one of the Receiver step attenuators is installed.

For redefined physical ports (see `[SENSe:]UDSParams<Pt>:PARam`), the respective measurement receiver (b-wave) is significant. E.g. if a receover step attenuator is installed for physical port 1, then an attenuation factor can be applied to the (redefined) port receiving b1.

Note:

- `INPut<PhyPt>:ATTenuation` is not channel-specific; the value is valid for **all channels**. Use `[SENSe<Ch>:]POWer:ATTenuation` to set or query a channel-specific attenuation value.
- In presence of External switch matrices all VNA ports have to be equipped with receiver step attenuator option.

Suffix:

<PhyPt> Physical port number; if unspecified the numeric suffix is set to 1

Parameters:

<Attenuation> Attenuation factor for the received wave.

Range: 0 dB, 10 dB, 20 dB, 30 dB. UP and DOWN increment/decrement the attenuation in 10 dB steps. The analyzer rounds any entered value below the maximum attenuation to the closest step.

*RST: 0 dB

Default unit: dB

Example: `INP2:ATT 10`

Set the step attenuator for the wave received at port 2 and for all channels to 10 dB. The waves at the other test ports are not affected.

`SENS1:POW:ATT? BREC`

Query the receiver step attenuator setting at port 2 and for channel no. 1. The response is `10`.

INSTrument[:SELect] <Channel>

Selects a channel `<Ch>` as active channel. To select a channel number > 4 use the generalized command `INSTrument:NSELect`.

Parameters:

<Channel> CHANnel1 | CHANnel2 | CHANnel3 | CHANnel4

Number of the channel to be activated. The channel must be created before using `CONFigure:CHANnel<Ch>[:STATe] ON`.

*RST: CHANNEL1

Example: `CONF:CHAN2:STAT ON; :INST CHANnel2`

Create channel no. 2 and select it as the active channel.

OUTPut<Chn>:DPORt <Port>

Selects a source port for the stimulus signal (drive port). The setting acts on the active trace. The effect of the drive port selection depends on the measurement parameter associated to the active trace:

- If an S-parameter $S_{<out><in>}$ is measured, the second port number index <in> (input port of the DUT = drive port of the analyzer) is set equal to the selected drive port: Drive port selection affects the measured quantity.
- If a wave quantity or a ratio is measured, the drive port is independent from the measured quantity:

Note: Use the equivalent command `[SENSe<Chn>:]SWEep:SRCPort` to address port numbers > 4.

Suffix:

<Chn> Channel number used to identify the active trace

Parameters:

<Port> PORT<no>

Physical port number.
In "set direction" only constants PORT1 to PORT4 can be used.

*RST: PORT1

Example: `CALC4:PAR:SDEF 'Ch4Tr1', 'A1'`
Create channel 4 and a trace named "Ch4Tr1" to measure the wave quantity a_1. The trace automatically becomes the active trace.
`OUTP4:DPOR PORT2`
Select drive port 2 for the active trace.

[SENSe<Ch>:]CORRection:CKIT:<ConnType>:<StandardType> <CalkitName>, <Standard>, <MinFreq>, <MaxFreq>, <DelayParam>, <Loss>, <C0>, <L0>, <C1>, <L1>, <C2>, <L2>, <C3>, <L3>, OPEN | SHORt

Defines the parameters of a calibration standard `<StandardType>` for a specified connector type `<ConnType>`. A particular physical standard can be selected by specifying the name of the calibration kit and its serial number. Depending on the standard type, only a subset of the parameters can be used; see Standard types and their parameters.

Note: If the specified cal kit does not exist, it is created with the specified calibration standard.

Suffix:

<Ch> Channel number. This suffix is ignored because calibration kits are channel-independent.

Parameters:

<ConnType> Connector type. One of the following identifiers:
`N50, N75`: N 50 Ω or N 75 Ω connectors
`PC7, PC35, PC292`: PC 7, PC 3.5 or 2.92 mm connectors
`USER<no>`: User-defined connectors UserConn1, UserConn2
`SMA`: User-defined connector type SMA ...
Note: This command only supports ZVR-compatible connector types. For general definitions, use `[SENSe:]CORRection:CKIT:<StandardType>`.

<StandardType> Standard type. For reflection standards, the first character denotes the gender, e.g.:
`FOPEN, MOPEN`: Open (f) or Open (m) standard.
For transmission standards, the first two characters denote the genders on both ends, e.g.:
`FFSNetwork, MFSNetwork, MMSNetwork`: Symm. network (ff), symm. network (mf) or symm. network (mm) standard.
For a complete list of standard types, refer to Standard types and their parameters.

Parameter list — String parameters to specify the configured standard (<CalkitName>, <StandardLabel>) and numeric parameters defining its properties. See Parameter list.

*RST: n/a

Example:
```
CORR:CKIT:N50:FOPEN 'ZV-Z21','',
0,1.8E+010,0.0151,0,0,0.22,-0.22,0.0022
```
Define the properties of the open (f) standard for the N 50 Ω connector type contained in the ZV-Z21 calibration kit: Assign a valid frequency range of 0 Hz to 18 GHz, an electrical length of 15.1 mm, 0 dB loss and define the polynomial coefficients of the fringing capacitance as 0 fF, 0.22 fF/GHz, –0.22 fF/(GHz)^2, 0.0022 fF/(GHz)^3.

[SENSe:]CORRection:CKIT:INSTall <CalKitFile>

Loads cal kit data from a specified R&S ZVR cal kit file.

Setting parameters:

<CalKitFile> — String parameter to specify the name and directory of the cal kit file to be loaded.
Note: The loaded file must be a R&S ZVR-specific cal kit file with the extension `*.ck`. VNA cal kit files (`*.calkit`) can be imported using the `MMEMory:LOAD:CKIT` command. Keysight cal kit files can be imported manually and converted into `*.calkit` files.

Example:
```
CORR:CKIT:INST 'C:\Users\Public\Documents
\Rohde-Schwarz\Vna\Calibration\Kits\ZCAN.ck'
```
Load the previously created R&S ZVR cal kit file `ZCAN.ck` from the default cal kit directory.
```
MMEM:STOR:CKIT 'ZCAN', 'C:
\Users\Public\Documents
\Rohde-Schwarz\Vna\Calibration\Kits\ZCAN.calkit'
```
Store the imported cal kit data to a VNA cal kit file `ZCAN.calkit` (assuming that the cal kit name stored in `ZCAN.ck` reads "ZCAN").

Usage: Setting only

[SENSe<Ch>:]CORRection:COLLect[:ACQuire] <Standard>[, <Dispersion>[, <Delay>]]

Starts a calibration measurement in order to acquire measurement data for the selected standards. The standards are reflection or transmission standards and must be connected to port 1 or 2 of the analyzer.

Tip: Use the generalized command `[SENSe<Ch>:]CORRection:COLLect[:ACQuire]:SELected` to obtain measurement data at arbitrary analyzer ports.

Suffix:

<Ch> Channel number of the calibrated channel.

Setting parameters:

<Standard> THRough | OPEN1 | OPEN2 | OPEN12 | SHORt1 | SHORt2 | SHORt12 | MATCh1 | MATCh2 | MATCh12 | NET | ATT | IMATch12 | REFL1 | REFL2 | SLIDe1 | SLIDe2 | SLIDe12 | LINe1 | LINe2 | LINe3 | LINe4 | LINe5 | LINe6 | M1O2 | O1M2 | OSHort1 | OSHort11 | OSHort12 | OSHort13 | OSHort2 | OSHort21 | OSHort22 | OSHort23 | M1S2 | S1M2 | UTHRough

Standard types: Through (between port 1 and 2), Open, Short, Match (MATCh12 and IMATch12 are synonymous), Symmetric Network (NET), Attenuation (ATT), Reflect, Sliding Match (SLIDe), Line1 (LINe1 and LINE are synonymous), Line2 to Line6 (esp. for TRL calibration), Match/Open (M1O2, O1M2), Match/Short (M1S2, S1M2), Offset Short (OSHort), Unknown Through (UTHRough).
The numbers in the parameter names denote the analyzer ports. Two numbers 12 mean that two separate calibrations are performed at ports 1 and 2. For Offset Short standards, the first number denotes the port (1 or 2), the second number denotes the number of the standard (1 to 3).

*RST: ON

<Dispersion> Optional status parameter for UTHRough standard:
OFF - unknown through standard is non-dispersive.
ON - unknown through standard is dispersive.

*RST: OFF

<Delay> Optional entry of delay time or phase for UTHRough standard:
<numeric> - entry of the delay time in ps (for non-dispersive standards) or of an estimate of the phase at the start frequency of the sweep in deg (for dispersive standards). See also background information for `[SENSe<Ch>:]CORRection:COLLect[:ACQuire]:SELected`.
AUTO - the analyzer determines the delay time or phase during the calibration sweep.

*RST: AUTO

Example: See `[SENSe<Ch>:]CORRection:COLLect:SAVE[:DUMMy]`

Usage: Setting only

[SENSe<Ch>:]CORRection:COLLect:METHod <CalType>

Selects a one-port or two-port calibration type for channel `<Ch>` at ports 1/2.

Tip: Use the generalized command `[SENSe<Ch>:]CORRection:COLLect:METHod:DEFine` to select the calibration type for arbitrary analyzer ports or a multiport calibration type.

Suffix:
<Ch> Channel number of the calibrated channel.

Parameters:
<CalType> FRTRans | PFRTrans | FTRans | PFTRans | RTRans | PRTRans | TOM | TSM | TRM | TRL | TNA | TOSM | ETOM | ETSM | FOPort1 | FOPort2 | FOPort12 | FOPTport | ROPTport | REFL1 | REFL2 | REFL12 | TPORt | UOSM

Calibration types, TOM, TRM, TRL, TNA, TOSM, Full One Port, One Path Two Port, Normalization (REFL1, REFL2 and REFL12 for one-port, FRTRans, FTRans, RTRans, and TPORT for two-port), TOSM with unknown through.
The numbers in the parameters denote the analyzer ports. Parameters for two-port calibration types contain no numbers because the command is only valid for ports 1 and 2.

Example: See `[SENSe<Ch>:]CORRection:COLLect:SAVE[:DUMMy]`

[SENSe<Ch>:]CORRection:COLLect:SAVE:DEFault

Generates a set of default system error correction data for the selected ports and calibration type. The default data set corresponds to a test setup which does not introduce any systematic errors; none of the measurement results acquired previously (`[SENSe<Ch>:]CORRection:COLLect[:ACQuire]`) is taken into account.

Tip: The main purpose of the default correction data set is to provide a dummy system error correction which you can replace with your own, external correction data. You may have acquired the external data in a previous session or even on an other instrument. If you want to use the external correction data on the analyzer, simply generate the default data set corresponding to your port configuration and calibration type and overwrite the default data. For details refer to the program example below.

Note: This command must be used in combination with the R&S ZVR-compatible commands `[SENSe<Ch>:]CORRection:COLLect:METHod` and `[SENSe<Ch>:]CORRection:DATA`. Use `[SENSe<Ch>:]CORRection:COLLect:SAVE:SELected:DEFault` if you want to use R&S ZNB-specific calibration commands or if you want to calibrate more than 2 ports.

Suffix:
<Ch> Channel number of the calibrated channel

Example:
`CORR:COLL:METH REFL1`
Select a one-port normalization at port 1 with an open standard as calibration type.
`CORR:COLL:SAVE:DEF`
Calculate a dummy system error correction for the normalization at port 1. The dummy system error correction provides the reflection tracking error term SCORR3.
`INIT:CONT OFF; :INIT; *WAI`
Stop the sweep to ensure correct transfer of calibration data.
`CORR:DATA? 'SCORR3'`
Query the dummy system error correction term. The response is a 1 (written as `1,0` for the real and imaginary part) for each sweep point (no attenuation and no phase shift between the analyzer and the calibration plane).
`CORR:DATA 'SCORR3',<ASCII_data>`
Replace the dummy system error correction term with your own correction data, transferred in ASCII format.
`INIT:CONT ON`
Restart the sweep in continuous mode.

Usage: Event

[SENSe<Ch>:]CORRection:COLLect:SAVE[:DUMMy]

Calculates the system error correction data from the acquired one or two-port measurement results (`[SENSe<Ch>:]CORRection:COLLect[:ACQuire]`), stores them and applies them to the calibrated channel `<Ch>`. To avoid incompatibilities, older system error correction data is deleted unless it has been transferred into a cal pool (`MMEMory:STORe:CORRection`).

This command is the R&S ZVR-compatible equivalent of `[SENSe<Ch>:]CORRection:COLLect:SAVE:SELected[:DUMMy]`. It must be used in combination with the R&S ZVR-compatible commands for calibration method and standard selection; see example below.

Suffix:
<Ch> Channel number of the calibrated channel.

Example:
`CORR:COLL:METH REFL1`
Select a one-port normalization at port 1 as calibration type.
`CORR:COLL OPEN1`
Measure an open standard connected to port 1 and store the measurement results of this standard.
`CORR:COLL:SAVE`
Calculate the system error correction data and apply it to the active channel.

Usage: Event

[SENSe<Ch>:]CORRection:DATA <ErrorTerm>, <Parameter>...

Writes or reads system error correction data for a specific channel `<Ch>` and calibration method (`[SENSe<Ch>:]CORRection:COLLect:METHod`). The analyzer test ports 1 or 2 are implicitly specified with the correction terms. The setting command can be used to transfer user-defined correction data to the analyzer; the query returns the current correction data set. ASCII or block data can be transferred, depending on the selected data transfer format (`FORMat[:DATA]`)

The sweep must be stopped to transfer calibration data; see program example for `[SENSe<Ch>:]CORRection:COLLect:SAVE:DEFault`.

Note: This command affects the active calibration of channel no. `<Ch>` or the factory calibration (if no channel calibration is active). For the factory calibration, the query form is allowed only (no change of factory calibration data).

Tip: Use the generalized command `[SENSe<Ch>:]CORRection:CDATa` to transfer calibration data for arbitrary analyzer ports. The analyzer provides a default calibration corresponding to a test setup which does not introduce any systematic errors; see `[SENSe<Ch>:]CORRection:COLLect:SAVE:DEFault`.

G and H matrices

The 7-term calibration types named Txx (e.g. TOM, TSM, TRM, TRL, TNA) are based on a network analyzer with two ports i and j, each equipped with a test receiver and a reference receiver. The system errors are described in terms of two "error two-ports" P_G and P_H:

- The error two-port P_G is assigned to port i of the analyzer. Its transmission matrix G describes how the system errors modify the outgoing and incident waves at port i:
$$\begin{bmatrix} b_i \\ a_i \end{bmatrix} = \begin{bmatrix} G_{11} & G_{12} \\ G_{21} & G_{22} \end{bmatrix} * \begin{bmatrix} m_{i\,ref} \\ m_{i\,test} \end{bmatrix}$$
- The error two-port P_H is assigned to port j of the analyzer. Its transmission matrix H describes how the system errors modify the measured incident and outgoing waves at port j:
$$\begin{bmatrix} a_j \\ b_j \end{bmatrix} = \begin{bmatrix} H_{11} & H_{12} \\ H_{21} & H_{22} \end{bmatrix} * \begin{bmatrix} m_{j\,test} \\ m_{j\,ref} \end{bmatrix}$$

In the two equations above, a and b denote the waves at the calibrated reference plane i and j (e.g. the input and output of the 2-port DUT). The m waves are the raw measured waves of test port i and j. The subscripts "ref" and "test" refer to the reference and test receivers, respectively. During the calibration the network analyzer acquires ratios of wave quantities, which leaves one of non-diagonal matrix elements of G or H as a free normalization factor. The network analyzer uses the normalization $H_{21} = 1$.

Suffix:
<Ch> Channel number of the calibrated channel

Parameters:
<ErrorTerm>

String parameters describing the different error terms, depending on the current calibration method; see table below. Each term contains one complex value (real and imaginary part) for each sweep point. The parameters must be transferred in full length. The following strings are allowed:
'SCORR1' - directivity at port 1
'SCORR2' - source match at port 1
'SCORR3' - reflection tracking at port 1
'SCORR4' - reserved for future extensions
'SCORR5' - load match at port 2
'SCORR6' - forward transmission tracking between port 1 and port 2
'SCORR7' - directivity at port 2
'SCORR8' - source match at port 2
'SCORR9' - reflection tracking at port 2
'SCORR10' - reserved for future extensions
'SCORR11' - load match at port 1
'SCORR12' - reverse transmission tracking between port 2 and port 1
'G11' ... 'G22' - G matrix elements; see above
'H22' - H matrix elements; see above
The error terms are dimensionless complex numbers.

*RST: n/a

<Parameter>

Example: See `[SENSe<Ch>:]CORRection:COLLect:SAVE:DEFault`

The different calibration types of the analyzer provide the following error terms:

Calibration type	Parameter in [SENSe<Ch>:]CORRection:COLLect:METHod	Available error terms (depending on port numbers)
One-port normalization (reflection) using an open standard	REFL1 REFL2 REFL12	'SCORR3' 'SCORR9' 'SCORR3' and 'SCORR9'
Full one port	FOPort1 FOPort2 FOPort12	'SCORR1' to 'SCORR3' 'SCORR7' to 'SCORR9' 'SCORR1' to 'SCORR3' and 'SCORR7' to 'SCORR9'
Two-port normalization	FTRans RTRans FRTRans	'SCORR6' 'SCORR12' 'SCORR6' and 'SCORR12'
One path two port	FOPTport ROPTport	'SCORR1' to 'SCORR3', 'SCORR6' 'SCORR7' to 'SCORR9', 'SCORR12'

Calibration type	Parameter in [SENSe<Ch>:]CORRection:COLLect:METHod	Available error terms (depending on port numbers)
TOSM	TOSM	'SCORR1' to 'SCORR12' (at present the isolation terms 'SCORR4' and 'SCORR10' are not included)
TOM, TSM, TRM, TRL, TNA	TOM | TRM | TRL | TNA	'DIRECTIVITY', 'SRCMATCH', 'REFLTRACK', 'LOADMATCH', 'TRANSTRACK' (for reading and writing) 'G11' ... 'G22' and 'H11', 'H12', 'H22' (for reading only; the 'H21' matrix elements are normalized to 1)

[SENSe<Ch>:]CORRection:OFFSet<PhyPt>:MAGNitude <OffsetLoss>

Defines the frequency-independent part (DC value) of the offset loss.

Tip: Use the `[SENSe<Ch>:]CORRection:LOSS<PhyPt>...` commands to define the complete set of loss offset parameters.
`[SENSe<Ch>:]CORRection:OFFSet<PhyPt>:MAGNitude` is equivalent to `[SENSe<Ch>:]CORRection:LOSS<PhyPt>:OFFSet`.

Suffix:
<Ch> Channel number of the offset-corrected channel

<PhyPt> Port number of the analyzer

Parameters:
<OffsetLoss> Frequency-independent part of the offset loss

Range: -200 dB to +200 dB
Increment: 0.0001 dB
*RST: 0 dB
Default unit: dB

[SENSe<Ch>:]FREQuency:CONVersion:MIXer:IFFixed <FixedFrequency>

Defines a fixed frequency and assigns it to the IF signal (for alternative commands see example below).

Suffix:
<Ch> Channel number.

Parameters:
<FixedFrequency> Fixed frequency.

Range: Depending on the instrument model.
Increment: 0.1 kHz
*RST: Minimum of the analyzer's frequency range, fMIN
Default unit: Hz

Example: `*RST; FREQ:CONV:MIX:IFF 1 GHz`
Reset the analyzer and specify a fixed frequency of 1 GHz, to be assigned to the IF signal.
`FREQ:CONV:MIX:FFIX?; FIX?`
Query the fixed frequency and the signal assignment using the alternative commands. The response is 1000000000;IF.

[SENSe<Ch>:]FREQuency:CONVersion:MIXer:LOFixed <FixedFrequency>

Defines a fixed frequency and assigns it to the LO signal (for alternative commands see example below).

Suffix:
<Ch> Channel number.

Parameters:
<FixedFrequency> Fixed frequency.

Range: Depending on the instrument model.
Increment: 0.1 kHz
*RST: Minimum of the analyzer's frequency range, fMIN
Default unit: Hz

Example: `*RST; FREQ:CONV:MIX:LOF 1 GHz`
Reset the analyzer and specify a fixed frequency of 1 GHz, to be assigned to the LO signal.
`FREQ:CONV:MIX:FFIX?; FIX?`
Query the fixed frequency and the signal assignment using the alternative commands. The response is 1000000000;LO.

[SENSe<Ch>:]FREQuency:CONVersion:MIXer:RFFixed <FixedFrequency>

Defines a fixed frequency and assigns it to the RF signal (for alternative commands see example below).

Suffix:
<Ch> Channel number.

Parameters:
<FixedFrequency> Fixed frequency.

Range: Depending on the instrument model.
Increment: 0.1 kHz
*RST: Minimum of the analyzer's frequency range, fMIN
Default unit: Hz

Example: `*RST; FREQ:CONV:MIX:RFF 1 GHz`
Reset the analyzer and specify a fixed frequency of 1 GHz, to be assigned to the RF signal.
`FREQ:CONV:MIX:FFIX?; FIX?`
Query the fixed frequency and the signal assignment using the alternative commands. The response is 1000000000;RF.

[SENSe<Ch>:]FREQuency:MODE <FreqSweep>

Selects the sweep type and defines which set of commands controls the stimulus frequency.

Tip: The command `[SENSe<Ch>:]SWEep:TYPE` provides a complete list of sweep types.

Suffix:
<Ch> Channel number

Parameters:
<FreqSweep> CW | FIXed | SWEep | SEGMent

Linear or logarithmic frequency sweep, depending on the selected spacing (`[SENSe<Ch>:]SWEep:SPACing LINear | LOGarithmic`). The frequency range is set via `[SENSe<Ch>:]FREQuency:STARt` etc.

Example:
```
FREQ:MODE CW
```
Activate a time sweep.
```
FREQ:CW 100MHz
```
Set the CW frequency to 100 MHz.

[SENSe<Ch>:]SEGMent<Seg>:CLEar

Deletes all sweep segments in the channel. The command is equivalent to `[SENSe<Ch>:]SEGMent<Seg>:DELete:ALL`.

Suffix:
<Ch> Channel number

<Seg> Sweep segment number. This suffix is ignored; the command deletes all segments.

Usage: Event

[SENSe<Ch>:]SEGMent<Seg>:OVERlap <Boolean>

Queries whether the analyzer supports overlapping sweep segments.

Suffix:
<Ch> Channel number. This suffix is ignored; the command is instrument-specific.

<Seg> Sweep segment number. This suffix is ignored; the command is instrument-specific.

Parameters:
<Boolean> ON | OFF: No effect.

*RST: ON. If used as a query, the command returns the information that overlapping sweep segments are supported (ON).

[SENSe<Ch>:]SWEep:SPACing <StimulusFreq>

Defines the frequency vs. time characteristics of a frequency sweep ("Lin Frequency" or "Log Frequency"). The command has no effect on segmented frequency, power or time sweeps.

Note: Use `[SENSe<Ch>:]SWEep:TYPE` to select sweep types other than "Lin Frequency" or "Log Frequency".

Suffix:
<Ch> Channel number

Parameters:
<StimulusFreq> LINear | LOGarithmic

The stimulus frequency is swept in equidistant steps over the frequency range. In a Cartesian diagram, the x-axis is a linear frequency axis.

Example: `FUNC "XFR:POW:S12"`
Activate a frequency sweep and select the S-parameter S_{12} as measured parameter for channel and trace no. 1.
`SWE:SPAC LOG`
Change to sweep type "Log Frequency".

[SENSe<Chn>:]FUNCtion[:ON] <SweepType>[, <arg1>[, <arg2>]]

Defines the sweep type and the measurement parameter in a single string.

Note: To select a measurement parameter without changing the sweep type, use `CALCulate<Ch>:PARameter:MEASure`. Use the other commands in the `CALCulate<Ch>:PARameter...` subsystem to create or delete traces and select measurement parameters.

Suffix:
<Chn> Channel number used to identify the active trace. If `[SENSe<Chn>:]FUNCtion[:ON]` is not used as a query, the number must be 1.

Parameters:
<SweepType> Single string parameter defining the sweep type and the parameter to be measured:
`<string> = "<sweep_type>:<parameter>".`

Range: See list of strings below.
*RST: "XFR:POW:S21"

<arg1> B1 | B2 | A1 | A2 | ABSa1 | ABSa2 | DCIN1 | DCIN2

<arg2> B1 | B2 | A1 | A2 | ABSa1 | ABSa2 | DCIN1 | DCIN2

Example: `CALC4:PAR:SDEF "Ch4Tr1", "S11"`
Create channel 4 and a trace named "Ch4Tr1" to measure the input reflection coefficient S_{11}. The trace automatically becomes the active trace.
`SENS4:FUNC?`
Check (query) the sweep type and measurement parameter of the active trace. The result is '`XFR:POW:S11`'.

The following keywords define the sweep type (see SCPI command reference: presentation layer):

`XFRequency`	Frequency sweep (Lin. Frequency/Log. Frequency/ Segmented Frequency)
`XPOWer`	Power sweep
`XTIMe`	Time sweep
`XCW?`	CW Mode sweep (output variable for query only)

The following keywords define the measurement parameter (see SCPI command reference: function name):

`POWer:S<Ptout><Ptin>`	S-parameter with output and input port number of the DUT, e.g. S11, S_{21}.
`POWer:RATio A<Ptout> | B<Ptin>, A<Ptout> | B<Ptin>` Output: `A<Ptout>/B<Ptin>`	Ratio, e.g. B2, A1 for b_2/a_1 drive Port 1
`POWer:A<Ptout>`	Wave quantity with stimulus port number of the analyzer, e.g. a_1.
`POWer:B<Ptin>`	Wave quantity with receive port number of the analyzer, e.g. b_2.
`POWer:Z<Ptout><Ptin>`	Matched-circuit impedance (converted Z-parameter) with output and input port number of the DUT, e.g. Z_{11}, Z_{21}.
`POWer:Y<Ptout><Ptin>`	Matched-circuit admittance (converted Y-parameter) with output and input port number of the DUT, e.g. Y_{11}, Y_{21}.
`POWer:KFACtor` `POWer:MUFactor<Lev>`	Stability factor K Stability factors μ_1 or μ_2
`VOLTage[:DC] DCIN1 | DCIN2` Output: `DC 1 V, DC 10 V`	DC Input 1 or 2

Note: The mnemonics POWer: and VOLTage: are not used in output strings.

SOURce<Ch>:POWer<PhyPt>:CORRection:COLLect[:ACQuire] <Sensor>

Initiates a source power calibration for the source port `<PhyPt>` using an external power meter no. 1 or 2. To initiate a source power calibration for arbitrary power meters, use the alternative commands listed in the program example below.

Note: The command cannot be used unless a power meter is connected via GPIB bus, USB or LAN interface and configured in the "External Power Meters" dialog.

Suffix:

<Ch> Calibrated channel number

<PhyPt> Calibrated port number

Setting parameters:

<Sensor> ASENsor | BSENsor

ASENsor – external power meter Pmtr 1.
BSENsor – external power meter Pmtr 2.

*RST: ASENsor

Example: `SOUR:POW3:CORR:COLL BSEN`
Perform a source power calibration for port 3 using power meter no. 2.
`SOUR:POW:CORR:PMET:ID 2`
Select power meter no. 2.
`SOUR:POW:CORR:ACQ PORT,3`
Perform a source power calibration for port 3 using the previously selected power meter no. 2.

Usage: Setting only

Manual operation: See "Start Cal Sweep" on page 598

TRACe:CLEar <MemTrace>

Deletes one of the memory traces `Mem<n>[Trc<m>]`, where n = 1, ... 8.

Setting parameters:

<MemTrace> MDATa1 | MDATa2 | MDATa3 | MDATa4 | MDATa5 | MDATa6 | MDATa7 | MDATa8

Identifier for the memory trace; see Table 7-19.

Range: MDATA<n> where <n> = 1 to 8.

Example: `SWE:POIN 20`
Create a trace with 20 sweep points, making the created trace the active trace of channel 1 (omitted optional mnemonic `SENSe1`).
`TRAC:COPY "Mem_Pt20",CH1DATA`
Copy the current state of the created trace to a memory trace named "Mem_Pt20". The memory trace is not displayed.
`DISP:WIND:TRAC2:FEED "MEM_PT20"`
Display the created memory trace in the active diagram area (diagram area no. 1).

Usage: Setting only

TRACe[:DATA][:RESPonse][:ALL]? <Response>

Returns the response values of the active data trace or memory trace (see Table 7-19).

Note: To read the response values of an arbitrary data or memory trace, use `CALCulate<Chn>:DATA`. To read the response values of a trace acquired in single sweep mode (`INITiate<Ch>:CONTinuous OFF`), use `CALCulate<Chn>:DATA:NSWeep:FIRSt?`.

Query parameters:

<Response> CH1Data | CH2Data | CH3Data | CH4Data | CH1Mem | CH2Mem | CH3Mem | CH4Mem | MDATa1 | MDATa2 | MDATa3 | MDATa4 | MDATa5 | MDATa6 | MDATa7 | MDATa8

Response data of the selected trace, see Table 7-19.
The data is transferred in the data format defined via `FORMat[:DATA]` and `FORMat:DEXPort:SOURce`. The unit is the default unit of the measured parameter; see `CALCulate<Ch>:PARameter:SDEFine`.

Example: `SWE:POIN 20`
Create a trace with 20 sweep points, making the created trace the active trace of channel 1 (omitted optional mnemonic `SENSe1`).
`CALC:FORM MLIN; :FORM ASCII; FORM:DEXP:SOUR FDAT`
Select the trace data format: linear magnitude values, ASCII format and formatted trace data (1 value per sweep point).
`TRAC? CH1DATA`
Query the 20 response values of the created trace according to the previous format settings.

Usage: Query only

TRACe[:DATA]:STIMulus[:ALL]? <Stimulus>

Returns the stimulus values of the active data trace or memory trace (see Table 7-19).

Note: To read the stimulus values of an arbitrary data or memory trace, use `CALCulate<Chn>:DATA:STIMulus?`

Query parameters:

<Stimulus> CH1Data | CH2Data | CH3Data | CH4Data | CH1Mem | CH2Mem | CH3Mem | CH4Mem | MDATa1 | MDATa2 | MDATa3 | MDATa4 | MDATa5 | MDATa6 | MDATa7 | MDATa8

Stimulus data of the selected trace, see Table 7-19.
The data is transferred in the data format defined via `FORMat[:DATA]`.

Example: `SWE:POIN 20`
Create a trace with 20 sweep points, making the created trace the active trace of channel 1 (omitted optional mnemonic `SENSe1`).
`TRAC:STIM? CH1DATA`
Query the 20 stimulus values of the created trace. In the default format setting, the data is returned as a comma-separated list of 10-digit ASCII values.

Usage: Query only

8 Programming examples

This chapter contains detailed programming examples on various tasks.

The syntax and use of all SCPI commands is described in Section 7, "Command reference", on page 911, where you will also find additional examples. For a general introduction to remote control of the analyzer refer to Section 6, "Remote control", on page 868. For an overview of special remote control features of the network analyzers refer to Section 6.3, "Basic remote control concepts", on page 882.

8.1 Basic tasks

This section presents detailed examples for programming tasks that almost every user will encounter when working with the R&S ZNB.

8.1.1 Typical stages of a remote control program

A typical remote control program comprises the following stages:

1. Performing the basic instrument settings
2. Adjusting the test setup
3. Initiating the measurement, command synchronization
4. Retrieving measurement results

Very often, steps 3 and 4 (or steps 2 to 4) must be repeated several times.

All example programs in this section have been developed and tested by means of the GPIB Explorer provided with the network analyzer. No extra programming environment is needed.

8.1.1.1 Basic instrument settings

Programming task: Adjust the basic network analyzer settings to your measurement tasks, optimizing the instrument for fast measurements.

Considerations for high measurement speed

The measurement speed depends on the sweep time but also on an efficient preparation of the instrument and on proper command synchronization. The following items should be kept in mind:

- For maximum speed the basic channel settings should be set while the sweep is stopped and with a minimum of sweep points. It is advisable to increase the number of points after all instrument settings have been performed, and to initiate the sweep after the test setup has been completed.
- Execution of the `INITiate[:IMMediate]` command is fastest in synchronized mode. Insertion of fixed waiting periods into the command sequence is possible but generally less efficient.
- The sweep time depends on several parameters; see below. In particular it is recommended to select the best set of sweep points, e.g. using the segmented sweep.

```
// Reset the instrument, switch off the measurement (after one sweep),
// reduce the number of sweep points.
*RST
INITiate1:CONTinuous OFF
SENSe1:SWEep:POINts 2
//
// Avoid a delay time between different partial measurements and before the
// start of the sweeps (is default setting).
SENSe1:SWEep:TIME:AUTO ON
TRIGger1:SEQuence:SOURce IMMediate
//
// Select the widest bandwidth compatible with your measurement.
SENSe1:BANDwidth:RESolution 10
//
// Adjust your sweep points to your measurement task, e.g. using a segmented sweep.
SENSe1:SEGMent...
```

8.1.1.2 Adjusting the test setup

In general the preparatives described above can be used for a series of measurements. In-between the measurements it is often necessary to change the test setup, e.g. to replace the DUT, change the connected ports, connect external devices etc.

8.1.1.3 Start of the measurement and command synchronization

Programming task: Start a measurement in single sweep mode. Wait until all single sweep data has been acquired before you proceed to the next stage of the measurement.

`INITiate<Ch>[:IMMediate][:DUMMy]` or `INITiate<Ch>[:IMMediate]:ALL` are used to start a single sweep or a group of single sweeps. These commands have

been implemented for overlapped execution. The advantage of overlapped commands is that they allow the program to do other tasks while being executed.

In the present example the sweep must be completed before measurement results can be retrieved. To prevent wrong results (e.g. a mix-up of results from consecutive sweeps) the controller must synchronize its operation to the execution of `INITiate<Ch>[:IMMediate]`. IEEE 488.2 defines three common commands (`*WAI`, `*OPC?`, `*OPC`) for synchronization.

// 1. Start single sweep, use *WAI

```
// *WAI is the easiest method of synchronization. It has no effect when sent
// after sequential commands.
// If *WAI follows INITiate<Ch>[:IMMediate]... (overlapped commands), the analyzer
// executes no further commands or queries until the sweep is terminated.
// *WAI does prevent the controller from sending other commands to the analyzer
// or other devices

//  Start single sweep in channel no. 1, wait until the end of the sweep
INITiate1:IMMediate; *WAI
<Continue program sequence>
```

// 2. Start single sweep, use *OPC?

```
// If *OPC follows INITiate<Ch>[:IMMediate]..., it places a 1 into the
// output queue when the sweep is terminated.
// An appropriate condition in the remote control program must cause the
// controller to wait until *OPC? returns one.
// The controller is stopped from the moment when the condition is set.

// Start single sweep in channel no. 1, indicate the end of the sweep
// by a 1 in the output queue.
INITiate1:IMMediate; *OPC?
// So far the controller may still send messages to other connected devices.

// Stop the controller until *OPC? returns one (program syntax depends
// on your programming environment).
<Condition OPC=1>
<Continue program sequence>
```

// 3. Start single sweep, use *OPC

```
// If *OPC follows INITiate<Ch>[:IMMediate]..., it sets the OPC bit in the ESR
// after the sweep is terminated.
// This event can be polled or used to trigger a service request of the analyzer.
// The advantage of *OPC synchronization is that both the controller and the
// analyzer can continue processing commands while the sweep is in progress.

// Enable a service request for the ESR
*SRE 32
// Set event enable bit for operation complete bit
```

```
*ESE 1
// Start single sweep in channel no. 1, set the OPC bit in the ESR
// after the sweep is terminated.
// The controller may still send messages, the analyzer continues to parse
// and execute commands.
INITiate1:IMMediate; *OPC

// Controller waits for service request from the analyzer
// (program syntax depends on your programming environment).
<Wait for service request>
<Continue program sequence>
```

8.1.1.4 Retrieving measurement results

Programming task: Read the results acquired in a single sweep.

// 1. Read single values (-> Markers)

```
// Markers are the most convenient tool for determining and retrieving single
// values on traces.
// The analyzer provides up to ten markers; see Markers and Limit Lines.
```

// 2. Read complete trace

```
// Select a trace format and read formatted trace data.
CALCulate1:FORMat MLINear  / Calculate the linear magnitude of z
CALCulate1:DATA? FDATa  / Read the formatted trace data
```

Use `CALCulate<Chn>:DATA:NSWeep:FIRSt?` to retrieve a particular trace within a group of sweeps.

8.1.2 Channel, trace and diagram handling

The following examples show you how to perform basic tasks related to channel and trace definition and to the display of traces in diagrams.

All example programs in this section have been developed and tested by means of the GPIB Explorer provided with the network analyzer. No extra programming environment is needed.

8.1.2.1 Several traces with equal channel settings

Programming task: Create up to four different traces with equal channel settings, assign the four 2-port standard S-parameters to the traces and display them in up to four diagrams.

Important remote control features for this program example

The following command sequence illustrates the structure of the remote commands discussed in section Basic Remote Control Concepts. In particular it shows that:

- A trace can be created and handled without being displayed.
- Traces are referenced by trace names. The active trace of a channel is often referenced by the channel suffix.
- Diagrams are referenced by a window suffix <Wnd>. An additional suffix <WndTr> in the DISPlay:WINDow<Wnd>:TRACe<WndTr>... commands numbers the different traces in a diagram.
- In remote control, it is possible to display the same trace in several diagrams.
- The analyzer provides several commands allowing a smooth transition between remote and manual control.

// 1. One channel, two traces, one diagram

```
// Reset the instrument, creating the default trace Trc1 in channel 1.
// The default measured quantity is the forward transmission S-parameter S21.
// The default format is dB Mag.
*RST

// Create a second trace in channel 1, assign the format Phase,
// and display the new trace in the same diagram.
// the trace becomes the active trace but is not displayed
CALCulate1:PARameter:SDEFine 'Trc2', 'S21'

// the trace is referenced by the channel suffix 1
CALCulate1:FORMat PHASe

// display the second trace, numbering it the second trace in diagram no. 1
DISPlay:WINDow1:TRACe2:FEED 'Trc2'
```

// Check the result on the local screen

```
// Go to local
SYSTem:DISPlay:UPDate ONCE
```

// 2. One channel, two traces, two diagrams

```
// Create a second diagram, assign Trc2 to the new area, and remove it
// from the first area.
DISPlay:WINDow2:STATe ON
DISPlay:WINDow2:TRACe2:FEED 'Trc2'
// Trc2 is now displayed in both diagrams
DISPlay:WINDow1:TRACe2:DELete
```

// Check the result on the local screen

```
// Go to local
SYSTem:DISPlay:UPDate ONCE
```

// 3. One channel, four traces, four diagrams

```
// Reset the instrument, add diagrams no. 2, 3, 4.
*RST; :DISPlay:WINDow2:STATe ON
DISPlay:WINDow3:STATe ON
DISPlay:WINDow4:STATe ON

// Assign the reflection parameter S11 to the default trace.
:CALCulate1:PARameter:MEASure 'Trc1', 'S11'

// Assign the remaining S-parameters to new traces Trc2, Trc3, Tr4;
// select the Smith chart format for the reflection parameters.
CALCulate1:FORMat SMITh  // Smith chart for the active trace Trc1
CALCulate1:PARameter:SDEFine 'Trc2', 'S21'
CALCulate1:PARameter:SDEFine 'Trc3', 'S12'
CALCulate1:PARameter:SDEFine 'Trc4', 'S22'
// Smith chart for the active trace Trc4, referenced by the channel number
CALCulate1:FORMat SMITh

// Display the new traces in diagrams no. 2 to 4.
DISPlay:WINDow2:TRACe2:FEED 'Trc2'
DISPlay:WINDow3:TRACe3:FEED 'Trc3'
DISPlay:WINDow4:TRACe4:FEED 'Trc4'
```

// Check the result on the local screen

```
// Go to local
SYSTem:DISPlay:UPDate ONCE
```

8.1.2.2 Several traces with different channel settings...

Programming task: Create three channels with 3, 1 and 2 traces, respectively, and display the traces in two diagrams.

Important remote control features for this program example

The following command sequence illustrates the structure of the remote commands discussed in section Basic Remote Control Concepts. In particular it shows that:

1. Channels are always referenced by a channel suffix.
2. Traces are referenced by trace names. The active trace of a channel is often referenced by the channel suffix.
3. Diagrams are referenced by a window suffix `<Wnd>`. An additional suffix `<WndTr>` in the `DISPlay:WINDow<Wnd>:TRACe<WndTr>...` commands numbers the different traces in a diagram.
4. The analyzer provides several commands allowing a smooth transition between remote and manual control.

// 1. Create all channels and traces

```
// Reset the instrument, creating the default trace Trc1 in channel 1.
// The default measured quantity is the forward transmission S-parameter S21.
// The default format is dB Mag.
*RST

// Create two more traces in channel 1, assigning a trace name and a measured
// quantity to each of them. Choose descriptive trace names (instead of the
// short default names used above).
CALCulate1:PARameter:SDEFine 'Impedance_trace', 'Z-S21'
// the trace becomes the active trace for channel 1 but is not displayed

CALCulate1:PARameter:SDEFine 'Admittance_trace', 'Y-S21'
// the trace becomes the active trace for channel 1

// Create channel 2 with one new trace, channel 3 with two new traces.
CALCulate2:PARameter:SDEFine 'Ratio_trace', 'B1/B2'
CALCulate3:PARameter:SDEFine 'Z_trace', 'Z21'
CALCulate3:PARameter:SDEFine 'Y_trace', 'Y21'
CALCulate3:PARameter:SELect 'Z_trace'
//  the trace created previously becomes the active trace for channel 3
// So far, only the default trace is displayed.
```

// Check the result on the local screen

```
// Go to local
SYSTem:DISPlay:UPDate ONCE
```

// 2. Create second diagram and display traces

```
DISPlay:WINDow2:STATe ON
DISPlay:WINDow1:TRACe2:FEED 'Admittance_trace'
DISPlay:WINDow1:TRACe3:FEED 'Y_trace'
DISPlay:WINDow2:TRACe1:FEED 'Impedance_trace'
DISPlay:WINDow2:TRACe2:FEED 'Ratio_trace'
DISPlay:WINDow2:TRACe3:FEED 'Z_trace'
```

// Check the result on the local screen

```
// Go to local
SYSTem:DISPlay:UPDate ONCE
```

// 3. Check and modify your configuration

```
// Query the traces in channel 1.
CALCulate1:PARameter:CATalog?
// The response is 'Trc1,S21,Impedance_trace,Z-S21,Admittance_trace,Y-S21'

// Query the reference level for the 'Z_trace'.
// The trace is referenced by its number in diagram no. 2.
DISPlay:WINDow2:TRACe3:Y:RLEVel?
```

```
// Change the display format for the 'Z_trace'. The trace is the active trace
// in channel 3, so it is referenced by the channel suffix 3.
// Update the display
CALCulate3:FORMat PHASe
// Update the display
SYSTem:DISPlay:UPDate ONCE
```

8.1.2.3 Markers and limit lines...

Programming task: Display two traces in a single diagram area, use markers to read results, and perform a limit check.

Important remote control features for this program example

The following command sequence illustrates the structure of the remote commands discussed in section Basic Remote Control Concepts. In particular it shows that:

1. Traces are referenced by trace names. The active trace of a channel is often referenced by the channel suffix. This simplifies the program syntax, e.g. in the commands for marker settings and for the limit check.
2. Diagrams are referenced by a window suffix `<Wnd>`. An additional suffix `<WndTr>` in the `DISPlay:WINDow<Wnd>:TRACe<WndTr>...` commands numbers the different traces in a diagram.
3. The analyzer provides several commands allowing a smooth transition between remote and manual control.

// 1. Create one channel, two traces, one diagram

```
// Reset the instrument, creating the default trace Trc1 in channel 1.
// The default measured quantity is the forward transmission S-parameter S21.
// The default format is dB Mag.
*RST

// Create a second trace in channel 1, assign the format Phase,
// and display the new trace in the same diagram.

// the trace becomes the active trace but is not displayed
CALCulate1:PARameter:SDEFine 'Trc2', 'S21'

// the trace is referenced by the channel suffix 1
CALCulate1:FORMat PHASe

// display the second trace, numbering it the second trace in diagram no. 1
DISPlay:WINDow1:TRACe2:FEED 'Trc2'
```

// Check the result on the local screen

```
// Go to local
SYSTem:DISPlay:UPDate ONCE
```

// 2. Marker settings

```
// Adjust the sweep range to consider an interesting segment of the trace and
// re-scale the diagram.
SENSe1:FREQuency:STARt 4.5 GHz; STOP 5.5 GHz
// in the autoscale command the trace is referenced by its number in the diagram
DISPlay:WINDow1:TRACe1:Y:SCALe:AUTO ONCE

// Select trace Trc1 as the active trace of the channel, define a reference
// marker and a delta marker.
// In the marker commands the active trace is referenced by the channel suffix.
CALCulate1:PARameter:SELect 'Trc1'

// the marker is set to the center of the sweep range
CALCulate1:MARKer1:STATe ON

// this command also creates the reference marker
CALCulate1:MARKer1:DELTa:STATe ON

// set the reference marker to the beginning of the sweep range
CALCulate1:MARKer1:REFerence:X 4.5 GHz

// Use the delta marker to search for the minimum of the trace and query the result.
// the query returns the stimulus and the response value at the marker position
CALCulate1:MARKer1:FUNCtion:EXECute MIN; RES?
```

// Check the result on the local screen

```
// Go to local
SYSTem:DISPlay:UPDate ONCE
```

Use the `CALCulate<Chn>:DATA...` commands to retrieve the complete trace; see Retrieving Measurement Results.

// 3. Limit lines and limit check

```
// Remove all markers and define a limit line for the active trace.
CALCulate1:MARKer1:AOFF
//  define an upper limit line across the entire sweep range
CALCulate1:LIMit:DATA 1, 4500000000, 5500000000, -5, -5
CALCulate1:LIMit:DATA 2, 4500000000, 5000000000, -10, -15
//  define two segments for the lower limit line
CALCulate1:LIMit:DATA 2, 5000000000, 5500000000, -15, -10

// Display the limit line and perform the limit check.
CALCulate1:LIMit:DISPlay:STATe ON
CALCulate1:LIMit:STATe ON; FAIL?
// if the trace failed the limit check; the response is 1
```

// Check the result on the local screen

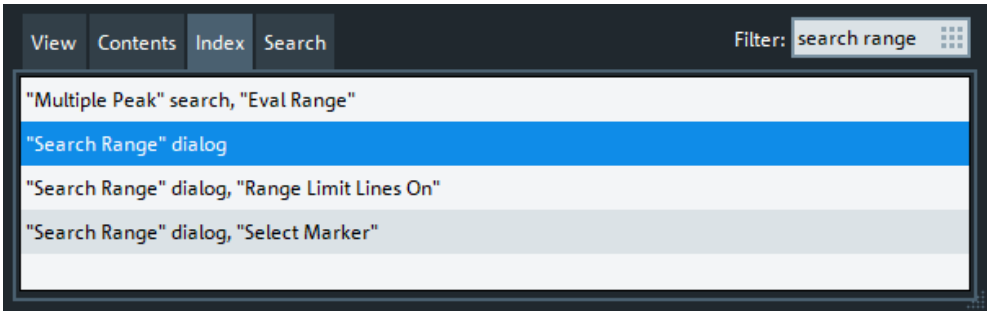
```
// Go to local
SYSTem:DISPlay:UPDate ONCE
```

8.2 Condensed programming examples

This section contains short program examples for select issues. The comments have been commented in concise style; for more detailed information on the commands refer to Section 7.3, "SCPI command reference", on page 917.

If the example you are looking for is not in this section, we suggest you to refer to the short command sequences in the reference section. Proceed as follows:

1. Find your subject in the help system, preferably using context-sensitivity (of the help system on your network analyzer) or the index:



2. Activate the link to the command description:

Remote command:
CALCulate<Chn>:MARKer<Mk>:FUNCtion:DOMain:USER[:RANGe]
CALCulate<Chn>:MARKer<Mk>:FUNCtion:DOMain:USER:STARt
CALCulate<Chn>:MARKer<Mk>:FUNCtion:DOMain:USER:STOP

3. A short example appears at the end of each command description:

Example:
```
CALC1:MARK1:FUNC:DOM:USER 2
```
Select the search range no. 2, assigned to marker no. 1 and trace no. 1.
```
CALC:MARK:FUNC:DOM:USER:STARt 1GHz
```
Set the start frequency of the search range to 1 GHz.
```
CALC:MARK:FUNC:DOM:USER:STOP 1.2GHz
```
Set the stop frequency of the search range to 1.2 GHz.

The command `SYSTem:DISPlay:UPDate` precedes some of the command scripts so that you can watch the progress of the script on the screen. For maximum performance, simply omit this command.

8.2.1 Path-independent remote control programs

The default directory for R&S ZNB user data is `C:\Users\Public\Documents\Rohde-Schwarz\Vna`. Other instruments may use different default directories. To make remote control programs compatible, it is recommended to define all paths relative to the default directory, to be set via `MMEMory:CDIRectory DEFault`.

// Select default directory, change to subdirectory (relative to default directory)

```
MMEMory:CDIRectory   DEFault
MMEMory:CDIRectory   'Traces'

MMEMory:STORe:TRACe 'Trc1', 'S21.s1p'
MMEMory:LOAD:TRACe 'Trc1', 'S21.s1p'
```

// Alternative, more compact definition

```
MMEMory:CDIRectory   DEFault
MMEMory:STORe:TRACe 'Trc1', 'Traces\S21.s1p'
```

You may also read the default path (`MMEMory:CDIRectory?`) and use the external RC program to build the complete paths.

8.2.2 Trace and diagram handling

The following sections provide examples for efficient channel and trace definition and convenient diagram handling.

8.2.2.1 Assigning channels, traces, and diagrams

The following example is a short version of Section 8.1.2, "Channel, trace and diagram handling", on page 1650.

// Reset the analyzer

```
*RST
:SYSTEM:DISPLAY:UPDATE ON
```

// Create a new trace for S21 with the name TrcDisp

```
:CALCULATE1:PARAMETER:SDEFINE "TrcDisp","S21"
// Display the trace in the display area 1
:DISPLAY:WINDOW1:TRACE9:FEED 'TrcDisp'
:DISPLAY:WINDOW1:TRACE9:DELETE
```

// List the traces assigned to a certain Channel

```
// format "<trace name>,<meas param>[,<trace name>,<meas param>...]"
:CALCULATE1:PARAMETER:CATALOG?
```

// Channel 4 does not exist, a new channel and trace is created

```
:CALCULATE4:PARAMETER:SDEFINE "Ch4Trc2","S22"
:CALCULATE4:PARAMETER:SDEFINE "Ch4Trc3","S33"
:CALCULATE4:PARAMETER:CATALOG?
```

// Select active traces for channel 4

```
:CALCULATE4:PARAMETER:SELECT "Ch4Trc2"
:CALCULATE4:PARAMETER:SELECT?
//:CALCULATE4:FORMAT POLAR
:CALCULATE4:PARAMETER:SELECT "Ch4Trc3"
//:CALCULATE4:FORMAT DB_LIN
:CALCULATE4:PARAMETER:SELECT?
:CALCULATE4:PARAMETER:SELECT "Ch4Trc2"
:CALCULATE4:FORMAT?
:CALCULATE4:PARAMETER:SELECT "Ch4Trc3"
:CALCULATE4:FORMAT?
```

// Create trace

```
:CALCULATE1:PARAMETER:SDEFINE "Trc2","S21"
:CALCULATE1:PARAMETER:SDEFINE "Trc3","S31"
:CALCULATE1:PARAMETER:CATALOG?
```

// Delete trace

```
:CALCULATE1:PARAMETER:DELETE "Trc2"
:CALCULATE1:PARAMETER:CATALOG?
```

// Assign a trace to a window = diagram, diagram 1 always exists

```
:DISPLAY:WINDOW1:TRACE2:FEED 'TrcDisp'
```

// Create diagram 2

```
:DISPLAY:WINDOW2:STATE?
:DISPLAY:WINDOW2:STATE ON
:DISPLAY:WINDOW2:STATE?
:DISPLAY:WINDOW2:TRACE6:FEED 'Ch4Trc2'
:DISPLAY:WINDOW2:TRACE2:FEED 'Trc1'
:DISPLAY:WINDOW2:TRACE3:FEED 'Ch4Trc3'
```

// Create traces: trace names are not case-sensitive

```
:CALCULATE4:PARAMETER:SELECT "Ch4Trc2"
:CALCULATE4:PARAMETER:SELECT?
:CALCULATE4:PARAMETER:SELECT "CH4TRC3"
:CALCULATE4:PARAMETER:SELECT?
:CALCULATE4:PARAMETER:SELECT "ch4trc2"
:CALCULATE4:PARAMETER:SELECT?
```

8.2.2.2 Memory traces

The following example shows how to save data to memory and work with memory traces.

// Reset the analyzer

```
*RST
:SYSTEM:DISPLAY:UPDATE ON

:SENSE1:SWEEP:POINTS 20
```

// Create a memory trace of the "active" trace (active for the parser !)

```
// the name of the created memory trace is "Mem2[Trc1]"
:TRACE:COPY MDATA2,CH1DATA
:SENSE1:FUNCTION:ON 'XFREQUENCY:POWER:S11'
:CALCULATE1:PARAMETER:CATALOG?

// Assign the memory trace to a window = diagram, diagram 1 always exists
:DISPLAY:WINDOW1:TRACE2:FEED 'Mem2[Trc1]'
```

// Create further memory traces and assign them to a window

```
:TRACE:COPY 'Mem3x[Trc1]',CH1DATA                // mixed parameters String, Char
:DISPLAY:WINDOW1:TRACE3:FEED 'Mem3x[Trc1]'
:TRACE:COPY MDATA4,CH1DATA
:DISPLAY:WINDOW1:TRACE4:FEED 'Mem4[Trc1]'
```

// Create a new normal trace on channel 1, assign it to a window

```
:CALCULATE1:PARAMETER:SDEFINE "Trc2","S22"
:DISPLAY:WINDOW1:TRACE5:FEED 'Trc2'
:CALCULATE1:PARAMETER:SELECT 'Trc2'    // now active for channel 1
:CALCULATE1:PARAMETER:SELECT?
:CALCULATE1:PARAMETER:CATALOG?
```

// Create memory trace for 'Trc2',

```
// The memory trace can be assigned to the diagram of the mother trace only
// (diagram 1)
:TRACE:COPY MDATA6,CH1DATA
:DISPLAY:WINDOW1:TRACE6:FEED 'Mem6[Trc2]'
```

// Create new diagram 2 and new channel and trace

```
:DISPLAY:WINDOW2:STATE ON
:CALCULATE2:PARAMETER:SDEFINE "Ch2Trc1","S22"
:CALCULATE2:PARAMETER:SELECT 'Ch2Trc1'    // now active for channel 2
:SENSE1:SWEEP:POINTS 21
:TRACE:COPY MDATA1,CH2DATA
:DISPLAY:WINDOW2:TRACE7:FEED 'Mem1[Ch2Trc1]'
```

```
:TRACE:DATA:STIMULUS? CH1DATA
:TRACE:DATA:RESPONSE? MDATA6
```

// Create a new channel 3 and a new trace

```
:CALCULATE3:PARAMETER:SDEFINE "Ch3Trc1","S21"
:CALCULATE3:PARAMETER:SELECT 'Ch3Trc1'    // now active for channel 3
:CALCULATE3:PARAMETER:SELECT? %'Ch3Trc1'
:SENSE1:SWEEP:POINTS 22
:TRACE:COPY MDATA8,CH3DATA
:TRACE:COPY MDATA7,CH3DATA
:DISPLAY:WINDOW2:TRACE1:FEED 'Mem8[Ch3Trc1]'
:TRACE:DATA:RESPONSE? MDATA7   // assigned to no diagram
```

// Copy with arbitrary trace names (no blanks allowed!)

```
:TRACE:COPY 'Trace_Name','Ch3Trc1'
:DISPLAY:WINDOW2:TRACE2:FEED 'Trace_Name'
:CALCULATE3:PARAMETER:SELECT 'Trace_Name'
:CALCULATE3:PARAMETER:SELECT?
:TRACE:COPY 'XYZ','Ch2Trc1'
:DISPLAY:WINDOW2:TRACE3:FEED 'XYZ'
:CALCULATE2:PARAMETER:SELECT 'XYZ'
:CALCULATE2:PARAMETER:SELECT?
:TRACE:COPY MDATA4 ,'Ch3Trc1'                // mixed parameters  Char, String
:DISPLAY:WINDOW2:TRACE4:FEED 'Mem4[Ch3Trc1]'
```

// Copy to existing memory traces = update trace data

```
:TRACE:COPY MDATA6,CH1DATA
:TRACE:COPY 'XYZ','Ch2Trc1'
:CALCULATE1:PARAMETER:CATALOG?
:CALCULATE2:PARAMETER:CATALOG?
:CALCULATE3:PARAMETER:CATALOG?
```

8.2.2.3 Trace mathematics

The following script shows how to define mathematical relations between traces.

// Reset the analyzer

```
*RST
:SYSTEM:DISPLAY:UPDATE ON
```

// Create memory trace on active trace of channel 1,

```
// assign the memory trace to a diagram
:CALCULATE1:MATH:MEMORIZE
:DISPLAY:WINDOW1:TRACE2:FEED 'Mem2[Trc1]'
```

// Define simple trace mathematics

```
:CALCULATE1:MATH:FUNCTION ADD
:CALCULATE1:MATH:FUNCTION?
// Trace mathematics off
:CALCULATE1:MATH:FUNCTION NORMAL
:CALCULATE1:MATH:FUNCTION?
*RST
```

// Create Trc2 in channel 1 and display it in diagram 1

```
:CALCULATE1:PARAMETER:SDEFINE "Trc2","S11"
:DISPLAY:WINDOW1:TRACE2:FEED 'Trc2'
```

// Create diagram 2 and Trc3 in the new channel

```
:DISPLAY:WINDOW2:STATE ON
:CALCULATE2:PARAMETER:SDEFINE "Trc3","S11"
:DISPLAY:WINDOW2:TRACE1:FEED 'Trc3'
```

// Select active traces for channels 1 and 2

```
:CALCULATE1:PARAMETER:SELECT "Trc1"
:CALCULATE2:PARAMETER:SELECT "Trc3"
```

// Create a memory trace on Trc1 and assign it to a diagram, same for TRC3

```
:TRACE:COPY MDATA7,CH1DATA
:DISPLAY:WINDOW1:TRACE3:FEED 'Mem7[Trc1]'
:TRACE:COPY MDATA8,CH2DATA
:DISPLAY:WINDOW2:TRACE2:FEED 'Mem8[Trc3]'
```

// Examples for Trace Mathematics

```
// Special operands Data and Mem
:CALCULATE1:MATH:SDEFINE "Data * Mem"
:CALCULATE1:MATH:STATE ON
// Constants
:CALCULATE1:MATH:SDEFINE "Pi * e * j"
// Functions
:CALCULATE1:MATH:SDEFINE "linMag (1) + dBMag (2) + Arg (3) + Re (4) + Im (5j)"
:CALCULATE1:MATH:SDEFINE "log (2) * ln (3) * Min (1, 2) * Max (2, 3)"
:CALCULATE1:MATH:SDEFINE "StimVal + asin (sin (3)) + acos (cos (4))
     + atan (tan (4))"
:CALCULATE1:MATH:SDEFINE "(Trc1 + 2) * 1.1"
:CALCULATE1:MATH:SDEFINE "(tRC1 + e) * Pi + STIMVAL - sin (1) + Min (TRC1, Trc1)"
// Imaginary unit j = sqrt (-1)
// j is no ordinary operand: 1j not 1 * j
// magnitude: 1, phase: 60 degrees
:CALCULATE1:MATH:SDEFINE "(1 + 3 ^ (1 / 2) * 1j) / 2"
:CALCULATE1:MATH:SDEFINE "sin (1) + ACOS (0.5)"
// 2 periods for sin (), ... when stop frequency 8 GHz = 8e9 Hz
```

```
:CALCULATE1:MATH:SDEFINE "sin (2 * 2 * Pi * StimVal / 8e9) "
:CALCULATE1:MATH:SDEFINE "cos (2 * 2 * Pi * StimVal / 8e9) "
:CALCULATE1:MATH:SDEFINE "tan (2 * 2 * Pi * StimVal / 8e9) "
:CALCULATE1:MATH:SDEFINE "Min (sin (6 * Pi * StimVal / 8e9),
     cos (6 * Pi * StimVal / 8e9))"
:CALCULATE1:MATH:SDEFINE "Max (sin (6 * Pi * StimVal / 8e9),
     cos (6 * Pi * StimVal / 8e9))"
:CALCULATE1:MATH:SDEFINE "Trc1 ^ 2"
:CALCULATE1:MATH:SDEFINE "Trc1 + Trc2 + Trc3"
:CALCULATE1:MATH:SDEFINE "(Trc1 + e) * Pi + Mem8[Trc3] + StimVal
     + Min (Trc1, Mem7[Trc1])"
:CALCULATE1:MATH:SDEFINE "tan (5 * 2 * Pi * StimVal / 8e9)"
```

8.2.2.4 Trace statistics

The following script shows how to create a trace, select an evaluation range and retrieve statistical results.

// Reset the analyzer

```
*RST
:SYSTEM:DISPLAY:UPDATE ON
```

// Create a new channel and trace

```
:CALCULATE2:PARAMETER:SDEFINE 'Trc2', 'S11'
:CALCULATE2:PARAMETER:SELECT  'Trc2'
:DISPLAY:WINDOW2:STATE ON
:DISPLAY:WINDOW2:TRACE1:FEED  'Trc2'
:SENSE1:SWEEP:TIME:AUTO ON
:SENSE2:SWEEP:TIME:AUTO ON
```

// Search full-span evaluation range, display statistical results

```
:CALCulate1:STATistics:DOMain:USER 0
:CALCulate2:STATistics:DOMain:USER 0
:CALCULATE1:STATISTICS ON
:CALCULATE2:STATISTICS ON
```

// Single sweep, global scope

```
:INITIATE:CONTINUOUS OFF
:INITIATE:IMMEDIATE:SCOPE ALL
:SENSE:SWEEP:COUNT 4
:INITIATE:IMMEDIATE; *WAI
```

// Calculate statistical results (also possible if info field is switched off)

```
:CALCULATE1:STATISTICS:RESULT? MEAN
:CALCulate1:STATistics:RESult? ELENgth
:CALCULATE1:STATISTICS:RESULT? ALL
```

// Modify evaluation range (is automatically confined to sweep range)

```
:CALCulate1:STATistics:DOMain:USER 1
:CALCulate1:STATistics:DOMain:USER:STARt   0 HZ
:CALCulate1:STATistics:DOMain:USER:STOP  100 GHZ
:CALCULATE1:STATISTICS:RESULT? MEAN

:CALCULATE1:STATISTICS:RESULT? MAX
```

8.2.2.5 Bandfilter search

The following example shows how to use markers for a bandpass or bandstop search.

// Reset the analyzer

```
*RST
:SYSTEM:DISPLAY:UPDATE ON

:SENSe1:FREQuency:STARt 1 GHZ
:SENSe1:FREQuency:STOP  6 GHZ
```

// Bandpass search ref. to max.

```
:CALCulate1:MARKer:FUNCtion:BWIDth:MODE   BPASs
:CALCulate1:MARKer:FUNCtion:BWIDth:MODE? %BPAS

// Measure single sweep, wait until complete sweep is finished
:INITiate:CONTinuous OFF
:INITiate; *WAI;
:CALCulate1:MARKer:FUNCtion:EXECute BFILter

// All markers OFF
:CALCulate1:MARKer:AOFF
```

// Bandpass search ref. to marker

```
:CALCulate1:MARKer1:STATe ON
:CALCulate1:MARKer1:X 3.0 GHz
:CALCulate1:MARKer:FUNCtion:BWIDth:MODE BPRMarker
:CALCulate1:MARKer:FUNCtion:BWIDth:MODE? %BPRM
:INITiate; *WAI;
:CALCulate1:MARKer:FUNCtion:EXECute BFILter
:CALCulate1:MARKer:AOFF
```

// Bandstop search ref. to max.

```
:CALCulate1:MARKer:FUNCtion:BWIDth:MODE   BSTop
:CALCulate1:MARKer:FUNCtion:BWIDth:MODE? %BST
:INITiate; *WAI;
:CALCulate1:MARKer:FUNCtion:EXECute BFILter
:CALCulate1:MARKer:AOFF
```

// Bandstop search ref. to marker

```
:CALCulate1:MARKer1:STATe ON
:CALCulate1:MARKer1:X 1.7 GHz
:CALCulate1:MARKer:FUNCtion:BWIDth:MODE   BsRMarker
:CALCulate1:MARKer:FUNCtion:BWIDth:MODE? %BsRM
:INITiate; *WAI;
:CALCulate1:MARKer:FUNCtion:EXECute BFILter
```

8.2.2.6 Creating diagrams

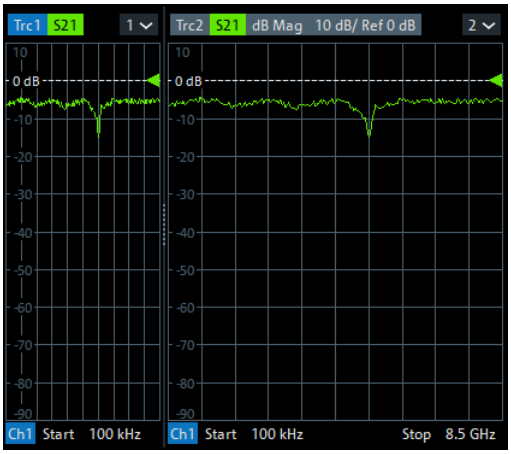
In the following example, remote control commands are used to position several diagrams on the screen. The remote control commands presented here extend the functionality of the "Display > Diagram" and "Display > Split" softtool tabs.

// Reset the analyzer

```
*RST
:SYSTEM:DISPLAY:UPDATE ON
```

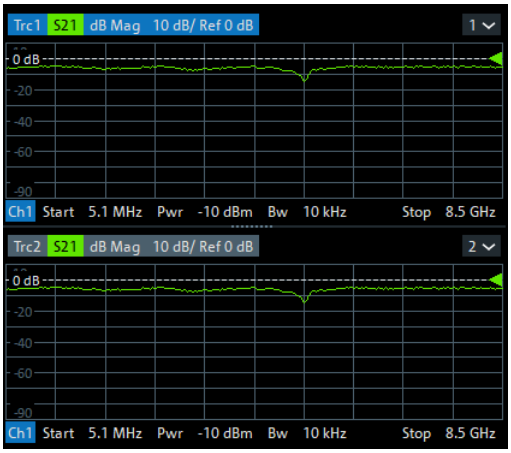
// Define and display a simple horizontal layout (two diagrams side by side)

```
:DISPlay:LAYout:DEFine 1, Horizontal, '1.00,0.30,0.70'
:DISPlay:LAYout:APPLy 1
```



// Define and display a simple vertical layout (two diagrams, one on top of the other)

```
:DISPlay:LAYout:DEFine 2, Vertical,   '1.00,0.50,0.50'
:DISPlay:LAYout:APPLy 2
```

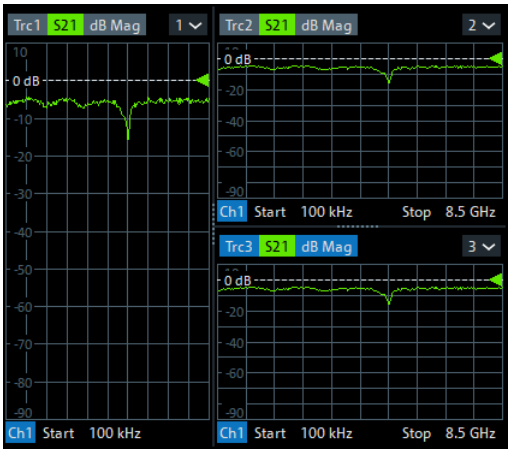


// Check the executable formats of the generated layouts

```
:DISPlay:LAYout:DEFine? 1 %(1,1,0.00,0.00,(1,2,1.00,1.00,[0.30,1.00],[0.70,1.00]))
:DISPlay:LAYout:DEFine? 2 %(1,1,0.00,0.00,(2,1,1.00,1.00,[1.00,0.50],[1.00,0.50]))
```

// Join the 2 layouts, display the nested layout

```
:DISPlay:LAYout:JOIN 1,2,2
:DISPlay:LAYout:APPLy 1
```



// Check the last applied (i.e. the joined) layout for the correct format

```
:DISPlay:LAYout:EXECute?
   %(1,1,0.00,0.00,(1,2,1.00,1.00,[0.30,1.00],
   %(1,1,0.70,1.00,(2,1,1.00,1.00,[1.00,0.50],[1.00,0.50])))))
```

// Set the layout format directly (horizontal, joined layout)

```
:DISPlay:LAYout:EXECute
   '(1,1,0.00,0.00,(1,2,1.00,1.00,[0.30,1.00],
     (1,1,0.70,1.00,(2,1,1.00,1.00,[1.00,0.50],[1.00,0.50])))))'
```

// Alternative direct definition of the joined layout as a vertical layout

```
:DISPlay:LAYout:DEFine 3, VERT, '0.3,1.0;0.7,0.5,0.5'
:DISPlay:LAYout:APPLy 3
:DISPlay:LAYout:EXECute?
   %(1,2,0.00,0.00,(1,1,0.3,1.00,[1.00,1.0]),(2,1,0.7,1.00,[1.00,0.5],[1.00,0.5]))
```

8.2.3 Using markers

The following example shows you how to define markers and use them to read trace values.

// Reset the analyzer

```
*RST
:SYSTEM:DISPLAY:UPDATE ON
```

// Initiate a single sweep

```
:INITiate1:CONTinuous OFF
:INITiate1:IMMediate;*WAI

// The following marker commands have the channel no. as a numeric suffix.
// The parameter belongs to a trace, the assignment channel -> trace
// is done via a "active" trace for each channel
:CALCULATE1:PARAMETER:SELECT 'Trc1'
```

// Marker ON / OFF

```
:CALCULATE1:MARKER1 ON
:CALCULATE1:MARKER1:STATE?
```

// Coupled Markers

```
// All markers belonging to channels with the same sweep type
// (FREQUENCY, TIME, POWER, CW FREQUENCY) are coupled/decoupled
:CALCULATE1:MARKER:COUPLED ON
```

// Marker Continuous / Discrete

```
:CALCULATE1:MARKER1:MODE CONTINUOUS
```

// Normal / Delta / Reference / Fixed Marker

```
:CALCULATE1:MARKER:AOFF           // all markers off
:CALCULATE1:MARKER1 ON
:CALCULATE1:MARKER1:X 1GHZ
:CALCULATE1:MARKER2 ON
:CALCULATE1:MARKER2:X 2GHZ
:CALCULATE1:MARKER3 ON
:CALCULATE1:MARKER3:X 3GHZ
:CALCULATE1:MARKER4:DELTA:STATE ON
```

```
:CALCULATE1:MARKER:REFERENCE ON
:CALCULATE1:MARKER:REFERENCE:X 5GHZ
:CALCULATE1:MARKER1:TYPE FIXED
```

// Query marker response values

```
:CALCULATE1:FORMAT MLINEAR
:CALCULATE1:MARKER1 ON
:CALCULATE1:MARKER1:X DEF
:CALCULATE1:MARKER1:FORMAT MLINEAR
// DataBase EMarkerFormat::LIN_MAG
:CALCULATE1:MARKER1:Y?
:CALCULATE1:MARKER1:FORMAT MLOGARITHMIC
//            .....        DB_MAG
:CALCULATE1:MARKER1:Y?
```

8.2.3.1 Marker search functions

The following example shows how to search for particular measurement points using markers.

// Reset the analyzer

```
*RST
:SYSTEM:DISPLAY:UPDATE ON
```

// Define marker and stimulus range

```
:CALCULATE1:MARKER1 ON
:SENSE1:FREQUENCY:START 1GHZ
:SENSE1:FREQUENCY:STOP 2GHZ
// Do the marker search in the format DB Magnitude
:CALCULATE1:FORMAT MLOGARITHMIC
```

// Define marker search ranges (stimulus range of the marker search)

```
// Range 0 is always the stimulus range of the trace (can't be changed)
:CALCULATE1:MARKER1:FUNCTION:DOMAIN:USER 0
:CALCulate1:MARKer1:FUNCtion:DOMain:USER:START?
:CALCulate1:MARKer1:FUNCtion:DOMain:USER:STOP?

// Range 1 (within the stimulus range)
:CALCULATE1:MARKER1:FUNCTION:DOMAIN:USER 1
:CALCulate1:MARKer1:FUNCtion:DOMain:USER:START 1.2GHZ
:CALCulate1:MARKer1:FUNCtion:DOMain:USER:STOP  1.8GHZ

// Range 2 (includes the stimulus range)
:CALCULATE1:MARKER1:FUNCTION:DOMAIN:USER 2
:CALCulate1:MARKer1:FUNCtion:DOMain:USER:START 0.8GHZ
:CALCulate1:MARKer1:FUNCtion:DOMain:USER:STOP  2.2GHZ
```

```
// Use range 0 (stimulus range of the trace)
:CALCULATE1:MARKER1:FUNCTION:DOMAIN:USER 0
```

// Select linear magnitude scale for diagram

```
:CALCULATE1:FORMAT MLINEAR
```

// Search for global minimum and maximum (MIN, MAX)

```
// (initial marker value may be inside or outside the marker search range)
:CALCULATE1:MARKER1:X 1.5GHZ
:CALCULATE1:MARKER:MIN
:CALCULATE1:MARKER1:X?

:CALCULATE1:MARKER1:X 1.5GHZ
:CALCULATE1:MARKER:MAX
:CALCULATE1:MARKER1:X?
```

// Minimum peak search functions

```
:CALCULATE1:MARKER1:FUNCTION:SELECT MINIMUM
:CALCULATE1:MARKER1:FUNCTION:SELECT?
// NEXT PEAK
:CALCULATE1:MARKER1:X 1.5GHZ
:CALCULATE1:MARKER:SEARCH:NEXT
:CALCULATE1:MARKER1:X?
// PEAK RIGHT
:CALCULATE1:MARKER1:X 1.5GHZ
:CALCULATE1:MARKER:SEARCH:RIGHT
:CALCULATE1:MARKER1:X?
// PEAK LEFT
:CALCULATE1:MARKER1:X 1.5GHZ
:CALCULATE1:MARKER:SEARCH:LEFT
:CALCULATE1:MARKER1:X?
```

// Maximum peak search functions

```
:CALCULATE1:MARKER1:FUNCTION:SELECT MAXIMUM
// Proceed as for minimum search
```

8.2.4 Data handling

The following sections provide examples for efficient sweep definition and data handling. Part of the functionality is not available in manual control.

8.2.4.1 Single sweep mode

The commands `CALCulate<Ch>:DATA:NSWeep...? SDATa, <Trace_Hist_Count>` retrieve the results of any sweep within a previously defined single sweep group. This means that, in single sweep mode, you can first measure a

specified number of sweeps (SENSe<Ch>:SWEep:COUNt <sweeps>) and then read any of the data traces acquired.

This feature has no equivalent in manual control where always the last data trace is displayed.

```
// Reset the analyzer
*RST
:SYSTEM:DISPLAY:UPDATE ON

// Create a second and third channel with new diagrams and traces.
:CALCULATE2:PARAMETER:SDEFINE "Trc2","S11"
:CALCULATE2:PARAMETER:SELECT  "Trc2"
:DISPLAY:WINDOW2:STATE ON
:DISPLAY:WINDOW2:TRACE1:FEED  'Trc2'

:CALCULATE3:PARAMETER:SDEFINE "Trc3","S11"
:CALCULATE3:PARAMETER:SELECT  "Trc3"
:DISPLAY:WINDOW3:STATE ON
:DISPLAY:WINDOW3:TRACE1:FEED  'Trc3'

// Select sweep time for the channels.
:SENSE1:SWEEP:TIME 1 S
:SENSE2:SWEEP:TIME 1 S
:SENSE3:SWEEP:TIME 1 S

// Enable single sweep mode for all channels so that channel-specific
// sweep count settings are used
:INITIATE:CONTINUOUS:ALL OFF

// Select single swep mode with channel-specific sweep count settings
// Set sweep counts and start measurement in all channels

:SENSE1:SWEEP:COUNT 1
:SENSE2:SWEEP:COUNT 2
:SENSE3:SWEEP:COUNT 3

:INITIATE1:IMMEDIATE; *WAI
:INITIATE2:IMMEDIATE; *WAI
:INITIATE3:IMMEDIATE; *WAI

// Select single swep mode with global sweep count settings

:SENSE:SWEEP:COUNT:ALL 2

:INITIATE1:IMMEDIATE; *WAI
:INITIATE2:IMMEDIATE; *WAI
:INITIATE3:IMMEDIATE; *WAI
```

8.2.4.2 Modeling a max hold function

The following example shows you how to emulate a max hold function.

// Reset the analyzer

```
*RST
:SYSTEM:DISPLAY:UPDATE ON
```

// Create a trace with the last extremum as memory trace.

```
:TRACe:COPY 'LastExtr', 'Trc1'

// Display this last extremum trace.
// Because it's a memory trace it must be displayed in the same diagram
// as the mother trace.
:DISPlay:WINDow1:TRACe2:FEED 'LastExtr'
:CALCulate1:MATH:SDEFine 'Max (Data, Mem)'
:CALCulate1:MATH:STATe ON
```

// Single sweep mode

```
:INITIATE:CONTINUOUS OFF

// Do a single sweep and update trace with the current extremum.
// This is the last extremum for the next sweep
:INITIATE:IMMEDIATE; *WAI
:TRACe:COPY:MATH 'LastExtr', 'Trc1'

// Loop over these 2 commands
:INITIATE:IMMEDIATE; *WAI
:TRACe:COPY:MATH 'LastExtr', 'Trc1'
:INITIATE:IMMEDIATE; *WAI
:TRACe:COPY:MATH 'LastExtr', 'Trc1'
```

// Continuous sweep mode

```
:INITIATE:CONTINUOUS ON
```

8.2.4.3 Retrieving the results of previous sweeps

The commands `CALCulate<Ch>:DATA:NSWeep...? SDATa, <Trace_Hist_Count>` retrieve the results of any sweep within a previously defined single sweep group. This means that, in single sweep mode, you can first measure a specified number of sweeps (`SENSe<Ch>:SWEep:COUNt <sweeps>`) and then read any of the data traces acquired.

This feature has no equivalent in manual control where always the last data trace is displayed.

// Reset the analyzer

```
*RST
:SYSTEM:DISPLAY:UPDATE ON
```

// Create a second channel with a second trace

```
:CALCULATE2:PARAMETER:SDEFINE "Trc2","S11"
:CALCULATE2:PARAMETER:SELECT  "Trc2"
:DISPLAY:WINDOW2:STATE ON
:DISPLAY:WINDOW2:TRACE1:FEED  'Trc2'
```

// Select active trace for the created channel 2. Adjust the number of sweep points.

```
:CALCULATE2:PARAMETER:SELECT "Trc2"
:SENSE1:SWEEP:POINTS 3
:SENSE2:SWEEP:POINTS 4
```

// Set sweep time and sweep count for the channels

(3 traces per single sweep in channel 1, 4 traces in channel 2)

```
:SENSE1:SWEEP:TIME 1 S
:SENSE2:SWEEP:TIME 1 S
:SENSE1:SWEEP:COUNT 3
:SENSE2:SWEEP:COUNT 4
```

// 1st Alternative: Reverse reading with command synchronization

Select single sweep mode and measure a single sweep group for channels no. 1 and 2

```
:INITIATE:CONTinuous:ALL OFF
:INITIATE:IMMEDIATE:ALL; *WAI
```

Read trace data (without history, i.e. the last trace acquired in each channel)

```
:CALCULATE1:DATA? SDATA
:CALCULATE2:DATA? SDATA
```

Read last and previous trace data in channels 1 and 2

```
:CALCULATE1:DATA:NSWEEP? SDATA, 1              // last trace data
:CALCULATE1:DATA:NSWEEP? SDATA, 3              // previous trace data
:CALCULATE2:DATA:NSWEEP? SDATA, 1              // last trace data
:CALCULATE2:DATA:NSWEEP? SDATA, 4              // previous trace data
```

// 2nd Alternative: Forward reading (no command synchronization necessary)

Select single sweep mode and measure a single sweep group for channels no. 1

```
:INITIATE1:CONTinuous OFF
:INITIATE1:IMMEDIATE
```

Read the first and the following trace data in channel 1

```
if (CALCULATE1:DATA:NSWEEP:COUNT? > 2)
:CALCULATE1:DATA:NSWEEP:FIRST? SDATA, 1              // first trace data
:CALCULATE1:DATA:NSWEEP:FIRST? SDATA, 3              // third trace data
```

8.2.4.4 Exporting S-parameters

The calibration defines which S-parameters are allowed to be exported to a Touchstone file. In the following example, a default (TOSM) calibration is created to make all S-parameters available.

// Reset the analyzer

```
*RST
:SYSTEM:DISPLAY:UPDATE ON

:SENSe1:CORRection:COLLect:METHod:DEFine 'Test', TOSM, 1, 2, 3, 4
:SENSe1:CORRection:COLLect:SAVE:SELected:DEFault
```

// Initiate a complete sweep

```
:INITiate1:CONTinuous OFF; :INITiate:IMMediate;*WAI
:MMEMory:STORe:TRACe:PORTs  1, 'ParserTouchstonePorts.s1p', COMPlex, 2
:MMEMory:STORe:TRACe:PORTs  1, 'ParserTouchstonePorts.s2p', COMPlex, 3, 2
:MMEMory:STORe:TRACe:PORTs  1, 'ParserTouchstonePorts.s4p', COMPlex, 1, 4, 3, 2
```

8.2.4.5 Parallel measurements on multiple DUTs

For background information, see Section 4.1.4.3, "Parallel measurements on multiple DUTs", on page 85.

Requires a R&S ZNBT, or a 4-port R&S ZNB with internal second source.

// Reset the analyzer

```
*RST
:SYSTEM:DISPLAY:UPDATE ON
```

// Define DUTs

```
// DUT 1
:SOUR:GRO1:PPORTs 1,2
:SOUR:GRO1:NAME 'DUT 1'

// DUT 2
:SOUR:GRO2:PPORTs 3,4
:SOUR:GRO2:NAME 'DUT 2'
```

// Create and display traces (all in the same channel)

```
// DUT 1 --> first diagram
:DISP:WIND1:STAT ON
:CALC1:PAR:SDEF 'Trc1','S11'
:DISP:WIND1:TRAC1:FEED 'Trc1'
:CALC1:PAR:SDEF 'Trc2','S22'
:DISP:WIND1:TRAC2:FEED 'Trc2'
:CALC1:PAR:SDEF 'Trc3','S21'
```

```
:DISP:WIND1:TRAC3:FEED 'Trc3'
:CALC1:PAR:SDEF 'Trc4','S12'
:DISP:WIND1:TRAC4:FEED 'Trc4'
// DUT 2 --> second diagram
:DISP:WIND2:STAT ON
:CALC1:PAR:SDEF 'Trc5','S33'
:DISP:WIND2:TRAC6:FEED 'Trc5'
:CALC1:PAR:SDEF 'Trc6','S44'
:DISP:WIND2:TRAC7:FEED 'Trc6'
:CALC1:PAR:SDEF 'Trc7','S43'
:DISP:WIND2:TRAC8:FEED 'Trc7'
:CALC1:PAR:SDEF 'Trc8','S34'
:DISP:WIND2:TRAC9:FEED 'Trc8'
```

// Save results

```
// Change format to Touchstone 2.0 to maximize speed
:MMEMory:STORe:TRACe:OPTion:FORMat F20
// Save data in complex format, with common target impedance of 50 Ω
:MMEM:STOR:TRAC:PORT  1, "dut1.s2p", COMPlex, CIMPedance, 1, 2
:MMEM:STOR:TRAC:PORT  1, 'dut2.s2p', COMPlex, CIMPedance, 3, 4
```

8.2.5 Calibration

The following programming examples are related to system error correction.

8.2.5.1 One and two-port calibration

The following example calibrates one or two analyzer ports.

// Reset the analyzer

```
*RST
:SYSTEM:DISPLAY:UPDATE ON
```

// Set cal kit as active kit for N50

```
:SENSE:CORRECTION:CKIT:N50:SELECT  'ZV-Z121'
```

// Select connectors for the ports

```
:SENSE1:CORRECTION:COLLECT:CONNECTION1 N50MALE
:SENSE1:CORRECTION:COLLECT:CONNECTION2 N50MALE

// Don't save the cal standard measurements with apply cal, i.e. with the commands
// :SENSE1:CORRECTION:COLLECT:SAVE  or
// :SENSe1:CORRection:COLLect:SAVE:SELected
// Instead, use the global, channel-independent setting:
:SENSe:CORRection:COLLect:ACQuire:RSAVe:DEFault OFF
```

// Full one port = OSM

```
// Select cal procedure
:SENSe1:CORRection:COLLect:METHod:DEFine     'Test SFK OSM 1', FOPORT, 1

// Measure Standards
:SENSe1:CORRection:COLLect:ACQuire:SELected  OPEN,  1
:SENSe1:CORRection:COLLect:ACQuire:SELected  SHORT, 1
:SENSe1:CORRection:COLLect:ACQuire:SELected  MATCH, 1

// Apply cal
:SENSe1:CORRection:COLLect:SAVE:SELected
```

// 2-port TOSM

```
// Select cal procedure
:SENSe1:CORRection:COLLect:METHod:DEFine     'Test SFK TOSM 12', TOSM, 1, 2

// Measure Standards
:SENSe1:CORRection:COLLect:ACQuire:SELected  THROUGH, 1, 2
:SENSe1:CORRection:COLLect:ACQuire:SELected  OPEN,  1
:SENSe1:CORRection:COLLect:ACQuire:SELected  SHORT, 1
:SENSe1:CORRection:COLLect:ACQuire:SELected  MATCH, 1
:SENSe1:CORRection:COLLect:ACQuire:SELected  OPEN,  2
:SENSe1:CORRection:COLLect:ACQuire:SELected  SHORT, 2
:SENSe1:CORRection:COLLect:ACQuire:SELected  MATCH, 2

// Apply calibration
:SENSe1:CORRection:COLLect:SAVE:SELected
```

// Save / load cal files

```
// Save calibration in calibration file pool
// C:\Users\Public\Documents\Rohde-Schwarz\Vna\Calibration\Data
// The filename in the following two commands must not contain the path!
:MMEMORY:STORE:CORRection 1, 'OSM1 TOSM12.cal'
// load cal file from calibration file pool
:MMEMORY:LOAD:CORRection  1, 'OSM1 TOSM12.cal'
```

8.2.5.2 Semi-automatic calibration with characterized Through

Assuming 3.5 mm (m) test ports and a single 2-port cal unit with matching connectors.

```
*RST
:SENSe:CORRection:COLLect:CONNection:PORTs ALL
:SENSe:CORRection:COLLect:CONNection:GENDers ALL
:SENSe:CORRection:COLLect:CONNection PC35male
// Select a manual calibration method that requires a characterized Through
:SENSe:CORRection:COLLect:METHod:DEFine '', TOSM, 1, 2
// Prepare automatic calibration of one port standards on ports 1, 2
:SENSe:CORRection:COLLect:AUTO:ASSignment1:DEFine 1, 1, 2, 2
// Configure a separate known THROUGH
```

```
:SENSe:CORRection:CKIT:THRough:SELect '3.5 mm Ideal Kit'
// Enforce a CalU initialization
:SENSe:CORRection:COLLect:AUTO:REPeat
// Measure CalU calibration data (O, S and M, but not T)
:SENSe:CORRection:COLLect:AUTO:ASSignment1:ACQUire
// Measure the THROUGH between ports 1 and 2 using the cal kit
:SENSe:CORRection:COLLect:ACQuire:SELected THROUGH, 1, 2
// Save the calibration
:SENSe1:CORRection:COLLect:SAVE:SELected
```

8.2.5.3 MultiCal (with calibration unit)

The following example demonstrates how to create multiple system error corrections on the same channel.

// Reset the analyzer

```
*RST

////////////////////////////////////////////////////////////////////////////////
```

// Connect to the cal unit

```
:SYSTem:COMMunicate:RDEVice:AKAL:ADDRess 'ZN-Z51::999002'

////////////////////////////////////////////////////////////////////////////////
```

// Prepare the multi cal

```
// create the first calibration:
// with full multi port and factory calkit
:SENSe1:CORRection:COLLect:AUTO:CONFigure FNPort, ''
// the first cal addresses test ports 2,3 and 4 and explicitly
// assigns them to cal unit ports 2,3 and 4, respectively
:SENSE1:CORRECTION:COLLECT:AUTO:ASSignment1:DEFine 2,2,3,3,4,4
// get the number of port assignments of the current calibration
// expected result: 1
:SENSE1:CORRECTION:COLLECT:AUTO:ASSignment:COUNt?
// explore port assignment 1
// expected result: 2,2,3,3,4,4
:SENSE1:CORRECTION:COLLECT:AUTO:ASSignment1:DEFine?
// create the second calibration:
// full one port and factory calkit
:SENSe1:CORRection:COLLect:AUTO:CONFigure FOPort, ''
// the second cal addresses test port 1
// and explicitly assigns it to cal unit port 1
:SENSE1:CORRECTION:COLLECT:AUTO:ASSignment2:DEFine 1,1
// get the number of port assignments of the current calibration
// expected result: 1
:SENSE1:CORRECTION:COLLECT:AUTO:ASSignment:COUNt?
// get the number of port assignments of all calibrations
```

```
// expected result: 2
:SENSE1:CORRECTION:COLLECT:AUTO:ASSignment:ALL:COUNt?
// explore port assignment 2
// expected result: 1,1
:SENSE1:CORRECTION:COLLECT:AUTO:ASSignment2:DEFine?


////////////////////////////////////////////////////////////////////////////////
```

// Acquire error correction data

```
:SENSE1:CORRECTION:COLLECT:AUTO:ASSignment1:ACQUire
:SENSE1:CORRECTION:COLLECT:AUTO:ASSignment2:ACQUire

////////////////////////////////////////////////////////////////////////////////
```

// Save the complete Multi-Cal

```
:SENSe1:CORRection:COLLect:AUTO:SAVE

////////////////////////////////////////////////////////////////////////////////
```

8.2.5.4 Saving and recalling error terms

The following examples show you how to perform a system error correction, save the acquired system error correction data to a file and reload them.

Performing a calibration, saving the error terms

// Reset the analyzer

```
*RST
:SYSTEM:DISPLAY:UPDATE ON
```

// Set frequency range

```
:SENSe1:FREQuency1:STARt 1GHz
:SENSe1:FREQuency1:STOP  4GHz
:SENSe1:SWEep:POINts 6
```

// Select calibration type: TOSM at ports 1 and 2

```
:SENSe1:CORRection:COLLect:METHod:DEFine  'Test SFK TOSM 12', TOSM, 1, 2

// Measure Standards
:SENSe1:CORRection:COLLect:ACQuire:SELected  THROUGH, 1, 2
:SENSe1:CORRection:COLLect:ACQuire:SELected  OPEN,  1
:SENSe1:CORRection:COLLect:ACQuire:SELected  SHORT, 1
:SENSe1:CORRection:COLLect:ACQuire:SELected  MATCH, 1
:SENSe1:CORRection:COLLect:ACQuire:SELected  OPEN,  2
```

```
:SENSe1:CORRection:COLLect:ACQuire:SELected  SHORT, 2
:SENSe1:CORRection:COLLect:ACQuire:SELected  MATCH, 2
```

// Apply calibration

```
SENSe1:CORRection:COLLect:SAVE:SELected
```

// Save error terms

```
:FORMAT REAL,32
@TRACEFILE:scorr1.dat
:CALCulate1:DATA? SCORr1
@TRACEFILE:scorr2.dat
:CALCulate1:DATA? SCORr2
@TRACEFILE:scorr3.dat
:CALCulate1:DATA? SCORr3
// We are omitting the isolation term, as it is not implemented
// @TRACEFILE:scorr4.dat
// :CALCulate1:DATA? SCORr4
@TRACEFILE:scorr5.dat
:CALCulate1:DATA? SCORr5
@TRACEFILE:scorr6.dat
:CALCulate1:DATA? SCORr6
@TRACEFILE:scorr7.dat
:CALCulate1:DATA? SCORr7
@TRACEFILE:scorr8.dat
:CALCulate1:DATA? SCORr8
@TRACEFILE:scorr9.dat
:CALCulate1:DATA? SCORr9
// We are omitting the isolation term, as it is not implemented
// @TRACEFILE:scorr10.dat
// :CALCulate1:DATA? SCORr10
@TRACEFILE:scorr11.dat
:CALCulate1:DATA? SCORr11
@TRACEFILE:scorr12.dat
:CALCulate1:DATA? SCORr1
```

Performing a new calibration, recalling the error terms

// Reset the analyzer

```
*RST
```

// Set frequency range

```
:SENSe1:FREQuency1:STARt 1GHz
:SENSe1:FREQuency1:STOP  4GHz
:SENSe1:SWEep:POINts 6
```

// Select calibration type: TOSM at ports 1 and 2

```
:SENSe1:CORRection:COLLect:METHod:DEFine  'XYZ', TOSM, 1, 2
```

// Generate a set of default correction data, switch on user calibration

```
:SENSe1:CORRection:COLLect:SAVE:SELected:DEFault
SENSE1:CORRECTION:STATE ON

// In the previous section, the following error terms were saved:
// SCORR1 Forward Directivity
// SCORR2 Forward Source Match
// SCORR3 Forward Reflection Tracking
// SCORR4 Forward Isolation
// SCORR5 Forward Load Match
// SCORR6 Forward Transmission Tracking
// SCORR7 Reverse Directivity
// SCORR8 Reverse Source Match
// SCORR9 Reverse Reflection Tracking
// SCORR10 Reverse Isolation
// SCORR11 Reverse Load Match
// SCORR12 Reverse Transmission Tracking

// Equivalences between the first parameter of CALculate:DATA
// and the first 3 Parameters of SENSe:CORRection:CDATa

// 'SCORR1'  'DIRECTIVITY', 1, 0 <Port 2> = 0 ignored
// 'SCORR2'  'SRCMATCH',    1, 0 <Port 2> = 0 ignored
// 'SCORR3'  'REFLTRACK',   1, 0 <Port 2> = 0 ignored
// 'SCORR4'  'ISOLATION',   1, 2
// 'SCORR5'  'LOADMATCH',   1, 2
// 'SCORR6'  'TRANSTRACK',  1, 2
// 'SCORR7'  'DIRECTIVITY', 2, 0 <Port 2> = 0 ignored
// 'SCORR8'  'SRCMATCH',    2, 0 <Port 2> = 0 ignored
// 'SCORR9'  'REFLTRACK',   2, 0 <Port 2> = 0 ignored
// 'SCORR10' 'ISOLATION',   2, 1
// 'SCORR11' 'LOADMATCH',   2, 1
// 'SCORR12' 'TRANSTRACK',  2, 1
```

// Set format for data transfer

```
:FORMAT REAL,32
```

// IMPORTANT: Stop sweep when loading error terms

```
:INITiate:CONTinuous OFF
```

// Recall error terms

```
// CALCulate:DATA is suitable for 2-port terms;
// use SENSe:CORRection:CDATa for more than 2 ports
:CALCulate1:DATA SCORR1,#@scorr1.dat
:CALCulate1:DATA SCORR2,#@scorr2.dat
```

```
:CALCulate1:DATA SCORR3,#@scorr3.dat
// We are omitting the isolation term, as it is not implemented on the ZVAB
//:CALCulate1:DATA SCORR4,#@scorr4.dat
:CALCulate1:DATA SCORR5,#@scorr5.dat
:CALCulate1:DATA SCORR6,#@scorr6.dat
:CALCulate1:DATA SCORR7,#@scorr7.dat
:CALCulate1:DATA SCORR8,#@scorr8.dat
:CALCulate1:DATA SCORR9,#@scorr9.dat
// We are omitting the isolation term, as it is not implemented on the ZVAB
//:CALCulate1:DATA SCORR10,#@scorr10.dat
:CALCulate1:DATA SCORR11,#@scorr11.dat
:CALCulate1:DATA SCORR12,#@scorr12.dat

:INITiate:CONTinuous ON
```

Using calibration data recorded previously

You can reuse any set of correction data that you acquired in earlier sessions on your analyzer. In general, you have to carry out the following steps:

1. Create a dummy correction data set and store it to a file.
2. Replace the dummy data with your correction data.
3. Reimport the correction data file and apply it to a channel.

See also the program examples for the following commands:

- `[SENSe<Ch>:]CORRection:COLLect:SAVE:SELected:DEFault` (dummy system error correction)
- `SOURce<Ch>:POWer<Pt>:CORRection:DEFault` (dummy source power calibration without external power meter)

8.2.5.5 Adapter removal

// define calibration method

```
SENSe1:CORRection:COLLect:METHod:DEFine 'Parser Test SFK', ARTosm, 1, 2
```

// Start with calkit-1 (e.g. 3.5 mm ideal Kit)

```
//adapter side port 1
SENSE1:CORRECTION:COLLECT:ACQuire:SELected OPEN,  1, ON
SENSE1:CORRECTION:COLLECT:ACQuire:SELected SHORT, 1, ON
SENSE1:CORRECTION:COLLECT:ACQuire:SELected MATCh, 1, ON

//non adapter side port 2
SENSE1:CORRECTION:COLLECT:ACQuire:SELected OPEN,  2, OFF
SENSE1:CORRECTION:COLLECT:ACQuire:SELected SHORT, 2, OFF
SENSE1:CORRECTION:COLLECT:ACQuire:SELected MATCh, 2, OFF
```

// Continue with Calkit-1 (e.g. N 50 Ohm ideal Kit)

```
//adapter side port 2
SENSE1:CORRECTION:COLLECT:ACQuire:SELected OPEN,  2, ON
SENSE1:CORRECTION:COLLECT:ACQuire:SELected SHORT, 2, ON
SENSE1:CORRECTION:COLLECT:ACQuire:SELected MATCh, 2, ON

//non adapter side port 1
SENSE1:CORRECTION:COLLECT:ACQuire:SELected OPEN,  1, OFF
SENSE1:CORRECTION:COLLECT:ACQuire:SELected SHORT, 1, OFF
SENSE1:CORRECTION:COLLECT:ACQuire:SELected MATCh, 1, OFF
```

// connect adapter between port 1 and 2

```
SENSE1:CORRECTION:COLLECT:ACQuire:SELected THRough, 1, 2
```

// save calibration

```
SENSe1:CORRection:COLLect:SAVE:SELected
```

8.2.5.6 Inline calibration

See Section 4.5.5.4, "Inline calibration", on page 179.

// Select an ICS

```
SYSTem:COMMunicate:RDEVice:AKAL:ADDRess 'ZN-Z30-02::100010'
```

// Query ICU addresses

```
SYST:COMM:RDEV:AKAL:ADDR:SUBM:ALL?
// Returns something like
// '[1]ZN-Z33-02::101325,[2]ZN-Z33-02::101325,[3]ZN-Z33-02::101339'
```

// Perform a complete UOSM calibration

```
SENSe1:CORRection:COLLect:AUTO:CPORt 1
SENSe1:CORRection:COLLect:AUTO:TEMPcomp OFF
SENSe1:CORRection:COLLect:AUTO:CONFigure FNPort, ''
SENSe1:CORRection:COLLect:AUTO:ASSignment:DELete:ALL
// let's assume ICU [i] is connected to VNA port i
SENSe1:CORRection:COLLect:AUTO:ASSignment1:DEFine 1, 1, 2, 2
// Step 1: OSM
SENSe1:CORRection:COLLect:AUTO:ASSignment1:ACQUire
// Step 2: Unknown Through
SENSe1:CORRection:COLLect:AUTO:UTHRough 1, 2
SENSe1:CORRection:COLLect:AUTO:SAVE
```

Repeat OSM only

```
SENSe1:CORRection:COLLect:AUTO:CONFigure FNPort, ''
SENSe1:CORRection:COLLect:AUTO:ASSignment:DELete:ALL
SENSe1:CORRection:COLLect:AUTO:ASSignment1:DEFine 1, 1, 2, 2
SENSe1:CORRection:COLLect:AUTO:REPeat // we will not repeat step 2
SENSe1:CORRection:COLLect:AUTO:ASSignment1:ACQUire
SENSe1:CORRection:COLLect:AUTO:SAVE
```

8.2.6 Mixer measurements

The following example is a remote implementation of the tasks the Mixer Presetting wizard performs at the GUI.

A power meter and a four-port calibration unit have to be connected and configured beforehand.

// Reset the analyzer

```
*RST
```

// Configure a linear frequency sweep

```
SENSe1:SWEEp:TYPE LINear       // Sweep type
SENSe1:SWEEp:POINts 201        // Number of points
SENSe1:FREQuency:STARt 500 MHZ // Start frequency
SENSe1:FREQuency:STOP 2.5  GHZ // Stop frequency
SOURce1:POWer1 0 DBM           // Base power
```

// Define the mixer

```
SENSe1:FREQuency:CONVersion:MIXer:STAGes 1             // Single mixer stage
SENSe1:FREQuency:CONVersion:MIXer:RFPort 1             // RF port = Port 1
SENSe1:FREQuency:CONVersion:MIXer:FUND RF              // RF @ channel frequency
SENSe1:FREQuency:CONVersion:MIXer:IFPort 2             // IF port = Port 2
SENSe1:FREQuency:CONVersion:MIXer:LOPort1 PORT,3       // LO1 port = Port 3
SENSe1:FREQuency:CONVersion:MIXer:FIXed LO1            // LO1 @ fixed frequency
SENSe1:FREQuency:CONVersion:MIXer:MFFixed LO1, 200 MHz // LO1 frequency
SOURce1:FREQuency:CONVersion:MIXer:PMFixed LO1, 7 DBM  // LO power
SENSe1:FREQuency:CONVersion:MIXer:TFRequency UCON      // IF = RF + LO
SENSe1:FREQuency:CONVersion MIXer                      // Activate mixer mode
```

// Perform an automatic SMARTerCal on ports 1, 2 and 3

```
SENSe1:CORRection:COLLect:PMETer:ID 1,1     // Power meter 1 @ port 1

//############ Connect VNA-ports 1,2,3 to Calu-ports 1,2,3
SENSe1:CORRection:COLLect:AUTO:PORTs:TYPE PFNPort, '', 1, 1, 2, 2, 3, 3
```

```
// The calibration is performed using the factory calibration

//############## Connect the powermeter to port 1
SENSe1:CORRection:COLLect:AUTO:POWer 1
```

// Perform source flatness cals on ports 1, 2 and 3

```
//################ Connect mixer: RF = port 1, IF = port 2, LO = port3
SOURce1:POWer1:CORRection:ACQuire PORT, 3  // Calibrate LO port
SOURce1:POWer1:CORRection:ACQuire PORT, 1  // Calibrate RF port
SOURce1:POWer1:CORRection:ACQuire PORT, 2  // Calibrate IF port
```

// Measure reflection at RF and IF ports

```
// Channel 1 with active trace 1 (= s21) already exists
// Trace 1 is displayed in diagram area 1
CONFigure:CHANnel1:NAME 'Ch_M'                // Rename channel 1
CONFigure:CHANnel1:TRACe:REName 'Conv'        // Rename trace 1

// Create and display trace 2 'RF_Refl' (S11)
CALCulate1:PARameter:SDEFine 'RF_Refl','S11' // Create trace 2
DISPlay:WINDow1:TRACe2:FEED 'RF_Refl'        // --> diagram area 1

// Create and display trace 3 'IF_Refl' (S22)
CALCulate1:PARameter:SDEFine 'IF_Refl','S22' // Create trace 3
DISPlay:WINDow1:TRACe3:FEED 'IF_Refl'        // --> diagram area 1
```

// Create a second channel for RF isolation measurements

```
CONFigure:CHANnel2 ON                         // Create channel 2
CONFigure:CHANnel2:NAME 'CH_RF'               // Rename channel 2
SENSe2:FREQuency:CONVersion FUNDamental       // Reset mixer mode

// RF isolation is measured with active LO (port 3)
// and with the same fixed LO frequency as in mixer channel 1
SOURce2:FREQuency3:CONVersion:ARBitrary:IFRequency 1, 1, 200 MHz, FIXed
SOURce2:POWer3:PERManent:STATe ON             // LO port: generator port

// Create and display trace 4 'RF_Isol' (S21)
CALCulate2:PARameter:SDEFine 'RF_Isol', 'S21' // Create trace 4
DISPlay:WINDow1:TRACe4:FEED 'RF_Isol'         // --> diagram area 1
```

// Create a third channel for LO leakage and feed-through

```
CONFigure:CHANnel3 ON                         // Create channel 3
CONFigure:CHANnel3:NAME 'CH_LO'               // Rename channel 3
SENSe3:FREQuency:CONVersion FUNDamental       // Reset mixer mode
```

```
// Use a fixed channel frequency, set to the LO frequency of mixer channel 1
SENSe3:FREQuency:STARt 200 MHz  // Start frequency =
SENSe3:FREQuency:STOP 200 MHz   //    Stop frequency = 200 MHz
SENSe3:SWEep:POINts 1           // One sweep point is enough !

// Create and display trace 5 'LO_Leak' (S13)
CALCulate3:PARameter:SDEFine 'LO_Leak', 'S13' // Create trace 5
DISPlay:WINDow1:TRACe5:FEED 'LO_Leak'         // --> diagram area 1

// Create and display trace 6 'LO_Thru' (S23)
CALCulate3:PARameter:SDEFine 'LO_Thru','S23'  // Create trace 6
DISPlay:WINDow1:TRACe6:FEED 'LO_Thru'         // --> diagram area 1
```

8.2.7 RFFE/GPIO interface programming

Requires hardware option R&S ZN-B15/-Z15 (see Section 4.7.20, "RFFE GPIO interface", on page 260).

Set GPIO Voltages directly

```
*RST
// Set GPIO voltages
:CONTROL:GPIO1:VOLTAGE:DEF +5V
// Write GPIO configuration into hardware
:CONTrol:GPIO:VOLTage:OUTPut
```

Sending RFFE commands directly

```
*RST
// General configuration of RFFE interface 2
:CONTROL:RFFE2:SETTings:FREQuency 50 KHZ
:CONTROL:RFFE2:SETTings:VOLTAGE:IO 2V
:CONTROL:RFFE2:SETTings:VOLTAGE:LOW 0V
:CONTROL:RFFE2:SETTings:VOLTAGE:HIGH 1.2V
// Send commands over RFFE interface 2
// Slave address: 0xA; command number: 0xBC; data: 0x0123
:CONTROL:RFFE2:COMMAND:DATA 'ABC0123'
:CONTROL:RFFE2:COMMAND:SEND
```

Reading a Product ID

This example assumes knowledge of the RFFE Specification of the MIPI® Alliance, in particular the "RFFE Supported Command Sequences".

```
*RST
// DUT Power on sequence
:CONTrol1:GPIO1:VOLTage 2.5V
:CONTROL1:GPIO:VOLTage:OUTPut
:CONTrol1:GPIO2:VOLTage 1.8V
:CONTROL1:GPIO:VOLTage:OUTPut
```

```
// Initialization of DUT with Slave Address SA = 0x0B:
// Write 0x38 at register address 0x1C.
// Therefore SA = 0xB; CMD = 0x5C; DATA = 0x38.
//          with CMD =    0010 0000b (see RFFE specification 'Register Write')
//                      | 0001 1100b (register address)
//                  =    0011 1100b
//                  =    0x5C
:CONTROL1:RFFE1:COMMAND:DATA 'B5C38'
:CONTROL1:RFFE1:COMMAND:SEND


// Read Product ID. Register address of product ID is noted in the DUT specification.
// This Example: Product ID is stored at address 0x1D.
// Therefore SA = 0xB; CMD = 0x7D;
//          with CMD =    0110 0000b (see RFFE specification 'Register Read')
//                      | 0001 1101b (register address)
//                  =    0111 1101b
//                  =    0x7D
:CONTROL1:RFFE1:COMMAND:DATA 'B7D'
:CONTROL1:RFFE1:COMMAND:SEND? 1      // Read 1Byte. Result is the Product ID
```

Configuring RFFE sweep sequencer (non-segmented sweep)

For non-segmented sweeps the RFFE device will be configured in preparation of every sweep.

```
*RST
// Enable GPIO 5 and RFFE interface 2 for Sweep Sequencer
:CONTROL:GPIO5:STATe ON
:CONTrol:RFFE2:SETTings:STATe ON

// Step 1
:CONTrol:SEQuence1:GPIO5:VOLTage 0V
:CONTrol:SEQuence1:RFFE2:COMMand:DATA '12345AF'
// Step 2
:CONTrol:SEQuence2:GPIO5:VOLTage -1V
// After step 2, wait for 3 ms
:CONTrol:SEQuence2:DELay 3ms
// Step 3
:CONTrol:SEQuence3:GPIO5:VOLTage 1V
// During sweep, output voltage will be 1V
```

Configuring RFFE sweep Sequencer (segmented sweep)

For segmented sweeps, the RFFE device will be configured in preparation of every segment.

```
*RST
// Define segmented sweep
:SENSE1:SEGMENT1:INSERT 1MHZ, 1.5MHZ, 111, -21DBM, 0.5S, 0, 10KHZ
:SENSE1:SEGMENT2:INSERT 2MHZ, 2.5MHZ, 222, -22DBM, 0.5S, 0, MAX
```

```
:SENSE1:SEGMENT3:INSERT 3MHZ, 3.5MHZ, 133, -23DBM, 0.5S, 0, 1KHZ
// Activate segemented sweep
:SENSe:SWEep:TYPE SEGMent

// Enable GPIO 4 and RFFE interfaces 1 for Sweep Sequencer
:CONTROL:GPIO4:STATe ON
:CONTrol:RFFE1:SETTings:STATe ON

// Set default voltage for GPIO Sweep Sequencer list
:CONTROL:GPIO4:VOLTAGE:DEF 0.5V

// Define command sequences per sweep segment
// Segment 1: A single step
:CONT:SEGM1:SEQ1:RFFE1:COMMAND:DATA '12311'
:CONT:SEGM1:SEQ1:GPIO4:VOLT 1.2V
// Segment 3: Two steps
:CONT:SEGM3:SEQ1:RFFE1:COMMAND:DATA '3EF11'
:CONT:SEGM3:SEQ2:RFFE1:COMMAND:DATA '3EF22'
```

8.2.8 Fixture modeling

We create and apply a fixture model for single-ended deembedding at ports 1 and 2 using the ISD tool.

Prepare the channel

```
*RST
:SENSe1:FREQuency:STARt MINimum
:SENSe1:FREQuency:STOP MAXimum
```

Perform a full two-port calibration at ports 1 and 2

For an example, see "// 2-port TOSM" on page 1674

Measure the test coupon

We are using a single-ended (2-port) "Symmetric 2x Thru" coupon (balanced). Connect it to ports 1 and 2.

```
:CALCulate:FMODel:ISD1:COUPon:TYPE SYMMetric2x
:CALCulate:FMODel:ISD1:COUPon:STATe 1
:CALCulate:FMODel:ISD2:COUPon:STATe 1
:CALCulate:FMODel:ISD3:COUPon:STATe 0
:CALCulate:FMODel:ISD4:COUPon:STATe 0
:CALCulate:FMODel:ISD:COUPon:MEASure
```

Measure the DUT together with the test fixture

Connect DUT + test fixture to ports 1 and 2.

```
:CALCulate:FMODel:ISD1:DUT:STATe 1
:CALCulate:FMODel:ISD2:DUT:STATe 1
:CALCulate:FMODel:ISD3:DUT:STATe 0
:CALCulate:FMODel:ISD4:DUT:STATe 0
:CALCulate:FMODel:ISD1:DUT:MEASure
:CALCulate:FMODel:ISD1:RUN:STATe 1
:CALCulate:FMODel:ISD2:RUN:STATe 1
:CALCulate:FMODel:ISD3:RUN:STATe 0
:CALCulate:FMODel:ISD4:RUN:STATe 0
:CALCulate:FMODel:ISD:RUN:RUN
```

Measure the DUT together with the test fixture

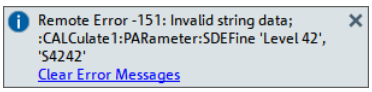
Run the ISD tool and apply the generated s2p files to port 1 (unswapped) and port 2 (swapped).

```
:CALCulate:FMODel:ISD1:RUN:STATe 1
:CALCulate:FMODel:ISD2:RUN:STATe 1
:CALCulate:FMODel:ISD3:RUN:STATe 0
:CALCulate:FMODel:ISD4:RUN:STATe 0
:CALCulate:FMODel:ISD:RUN:RUN
```

9 Error messages and troubleshooting

9.1 Errors during firmware operation

An error generally causes the analyzer to display a tooltip across the lower part of the screen. The tooltip provides a textual description of the error, e.g.:



The errors can be divided into four categories:

- Remote errors (SCPI errors) can occur during the execution of a remote control program. They include an error code, followed by the short description of the error. Remote errors are specified and described in the SCPI standard; they are cleared upon *CLS.
- Software errors (setting errors) can occur, e.g., if numeric entries in an analyzer dialog are incompatible with each other or with the current analyzer state. These errors are self-explanatory and easy to correct.
- Hardware errors indicate an incorrect hardware state. Some of the hardware errors cause the instrument to be switched off to avoid damage. Hardware errors with possible causes and remedies are listed in the following sections.
- Exceptions indicate anomalous or exceptional events that were not properly handled by the R&S ZNB firmware.

Troubleshooting SCPI errors

A misspelled command header causes SCPI error `-113, "Undefined header;..."`; a misspelled parameter causes `SCPI error -141, "Invalid character data;..."`. The GPIB explorer provides a list of all supported commands and their character data parameters; see Section 6.1.3, "GPIB Explorer", on page 870.

Hardware error categories

Hardware errors can be detected at various stages of the start-up or measurement procedure.

- Configuration errors occur on start-up of the analyzer, e.g. if a hardware module or configuration file cannot be detected. Configuration errors cause an entry in the error log (System > [Setup] > "Setup" > "Info" > "Error Log").
- Asynchronous errors can occur at any time while the analyzer is operating. The analyzer is checked periodically for asynchronous errors.
- Measurement errors are due to inadmissible hardware settings and states during the measurement process.

9.1.1 Asynchronous errors

Asynchronous errors can occur at any time while the analyzer is operating. The analyzer is checked periodically for asynchronous errors. Many of these errors also cause an entry in the status reporting system.

Error	Description	Remedy	Bit no.*)
Instrument temperature is too high	The analyzer detects that the instrument temperature is too high. After three warnings, the analyzer is shut down.	Reduce ambient temperature, keep ventilation holes of the casing unobstructed.	7
Receiver overload protection tripped	The analyzer detects an excessive input level at one of the ports. If this condition persists, all internal and external generators are switched off ("RF Off").	Reduce RF input level at the port. Check amplifiers in the external test setup.	3
Reference frequency lock failure	With external reference signal (System – External Reference active) or internal high precision clock, the reference oscillator is phase locked to a 10 MHz signal. The message appears when this phase locked loop (PLL) fails.	For external reference: check frequency and level of the supplied reference signal.	1
Converter clock frequency lock failure	The clock generator for the AD converter clock is phase locked to the reference oscillator. The message appears when this PLL fails.	–	–
OCXO cold	With OCXO option R&S ZNB-B4, the oven temperature is too low.	Wait until the oven has been heated up	8
Internal communication error	A severe unexpected disturbance was observed (e.g. due to ESD). The firmware resets all hardware components to ensure proper instrument functioning and restores the user settings to the state before the event. Finally, it indicates the foregone hardware reset by an information popup "Hardware communication problem detected. Hardware was reset. Measurement data of this sweep may be incorrect. Please contact Rohde & Schwarz if this problem persists."	If this error is raised repeatedly, please contact Rohde & Schwarz service.	6

*) The following bits in the "STATus:QUEStionable:INTegrity:HARDware" on page 899 register are set when the error occurs.

9.1.2 Errors during measurement

The following errors are due to inadmissible hardware settings and states during the measurement process. Some of the errors also cause an entry in the status reporting system.

Error	Description	Remedy	Bit no.*)
Unstable level control at port <i>	The analyzer detects an excessive source level at one of the ports. The signal is turned off and the sweep halted.	Check signal path for the wave a_i, especially check external components. Then press "Channel > Sweep > Sweep Control > Restart Sweep".	9
Problem concerning external generator Gen<i>	An external generator has been configured, however, it cannot be controlled or provides error messages. If several generators cause problems, the lowest number is indicated.	Check whether the generator is properly connected and switched on. Check the GPIB address; exclude address conflicts when using several external generators or other equipment.	10
Problem concerning external power meter or DC measurement device <i>	An external power meter or DC measurement device has been configured, however, it cannot be controlled or provides error messages. If several devices cause problems, the lowest number is indicated.	Check whether the external device is properly connected and switched on. Check its address and resolve address conflicts (with external devices of any type).	11
Time grid too close	The sweep points for a time sweep are too close, the analyzer cannot process the measurement data until the next sweep point starts.	Increase stop time, reduce no. of points, increase IF bandwidth. If possible reduce number of partial measurements, e.g. by restricting the number of ports measured.	12
Overload at DC MEAS <range>	The input voltage at one of the DC input connectors on the rear panel is too high.	Reduce input voltage.	13
Port <i> output power unleveled	The level control is unsettled or unstable, possibly due to an external disturbing signal.	Change generator level at the port; check external components.	2

*) The following bits in the `STATus:QUEStionable:INTegrity:HARDware` register are set when the error occurs.

9.2 Errors during firmware installation/update

During firmware installation or update, if the installer encounters an error, the update is canceled and the analyzer firmware is rolled back to its previous state.

Currently there is only one exception to this rule, which was implemented to recover gracefully from a failed FPGA update:

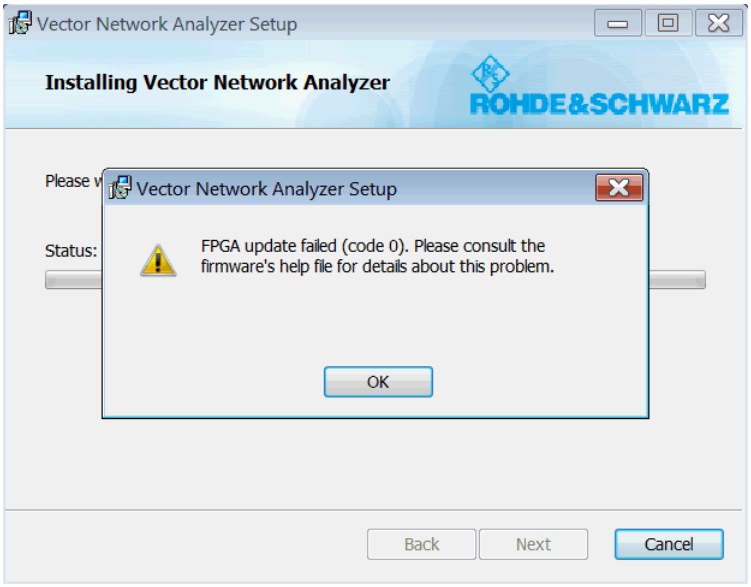
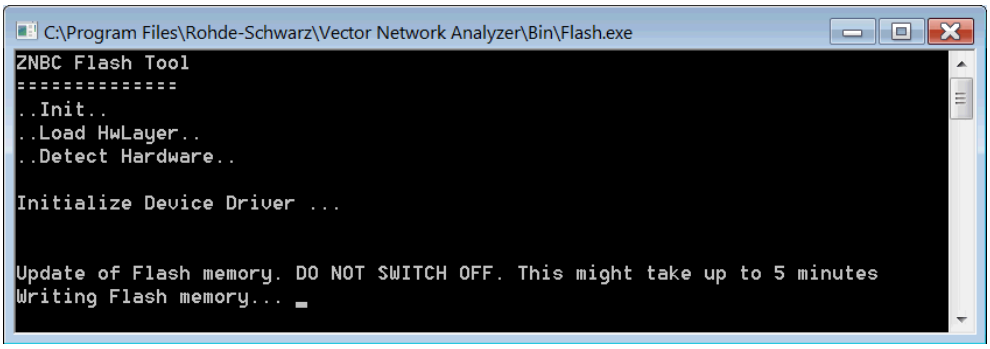


Figure 9-1: FPGA update failure

In this case, the installation proceeds but requires additional steps to perform the failed FPGA update manually.

No matter if the manual FPGA update described below is successful or not, write down the error code and contact our support.

1. Reboot the analyzer.
2. Exit the firmware.
3. Run the flash utility located at `C:\Program Files\Rohde-Schwarz\Vector Network Analyzer\ZNB\Bin\flash.exe`.



4. Wait until the process has completed.
5. Turn off the analyzer (a reboot is not sufficient).

   If successful, this procedure brings the analyzer back to working condition.

9.3 Collecting information for technical support

If you encounter problems that you cannot solve yourself, contact your Rohde & Schwarz support center, see Section 9.4, "Contacting customer support", on page 1692. Our support center staff is optimally trained to assist you in solving problems.

The support center finds solutions more quickly and efficiently if you provide them with information on the instrument and an error description.

Creating Windows event log Files

Windows records important actions of applications and the operating system in event logs. You can create event log files to summarize and save the existing event logs. If there is a problem with your product, you can send these files to Rohde & Schwarz.

To create Windows event log files

1. In the search field, enter *event Vvewer*.
2. Select "Open".
3. In the console tree on the left, select and expand "Windows Logs".
4. In the context menu of each subsection, select "Save All Events As...".

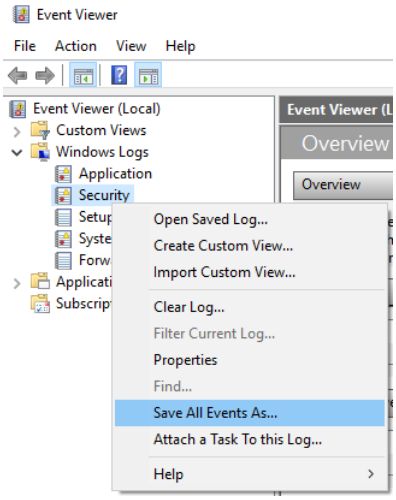


Figure 9-2: Event Viewer

5. Enter a filename.
6. Select "Save".

Obtaining information from the R&S ZNB firmware

The instrument generates error messages which are usually sufficient for you to detect the cause of an error and find a remedy. Error message types are described in Section 9, "Error messages and troubleshooting", on page 1687.

In addition, the system **"Info" dialog** offers valuable troubleshooting information. This dialog can be opened via the "Info" button in the System – [Setup] > "Setup" softtool tab.

For details on the available information, see Section 5.17.1.3, "Info dialog", on page 826.

The required troubleshooting information can also be generated in the system "Info" dialog. **"Create R&S Support Information"** collects the following information:

- Setup, option, and hardware information, error log, and selftest results.
  This information can also be retrieved using the "Save" function of the "Info" dialog.
- Current eeprom data.
- A screenshot of the VNA display.
- The latest 5 exception dumps.

The resulting `*.zip` file is written to the report directory `C:\Users\Public\Documents\Rohde-Schwarz\Vna\Report`. Its file name contains the current date and time, e.g. `report_20200228_1658.zip`.

9.4 Contacting customer support

Technical support – where and when you need it

For quick, expert help with any Rohde & Schwarz product, contact our customer support center. A team of highly qualified engineers provides support and works with you to find a solution to your query on any aspect of the operation, programming or applications of Rohde & Schwarz products.

Contact information

Contact our customer support center at www.rohde-schwarz.com/support, or follow this QR code:

Figure 9-3: QR code to the Rohde & Schwarz support page

10 File and data management

This section covers the functions the instrument provides for managing user data.

User data management typically involves locating the data under consideration and using standard Windows techniques for handling files and Windows event logs.

- Accessing, retrieving and handling data 1693
- Storage locations 1695

10.1 Accessing, retrieving and handling data

Depending on the origin and content, this description distinguishes between the following data types:

- Operational data
- Non-operational data

Operational data (data for intended use)

The instrument uses and creates this data because of its intended use and according to the settings and configurations you have made.

For example, see:

- Section 4.1.1, "Global (persistent) settings", on page 79
- Section 4.1.2, "Recall sets", on page 80
- Section 4.4.2, "Trace files", on page 143

You can access, retrieve and delete generated data using standard Windows file and event log management.

Recall sets can be serialized to a human-readable XML file format (`*.znxml`).

Non-operational data (usage data)

The R&S ZNB creates this data during and through its use. Such data is collected e.g. for troubleshooting, to help our customer support center find solutions quickly, or if you have purchased and activated the Health and Utilization Monitoring Service (HUMS).

Logging data is created continuously and in real-time, and is persisted on the instrument. You can export it to a remote server, for example via a network connection.

Table 10-1: Persistent non-operational data generated by the instrument

Type of data	Description	Data generator
User authentication	Windows login attempts	Windows event logging
Security and access control	Security events	Windows event logging
Network and communication	Network-related events	Windows event logging

Type of data	Description	Data generator
Software and firmware update	Windows update: Windows components and included third party components like drivers and applications	Windows event logging
	Firmware setup: • Temporary files in the `%TEMP%` directory of the Windows user running the update • Entries in `C:\Windows\INF\setupapi.dev.log` • Error logs in `%PROGRAMDATA%\Rohde-Schwarz\Vna\Log`	Installer & Windows SetupAPI
	HUMS: Logs in `C:\Windows\Temp`	Installer
Diagnostics and troubleshooting	Windows system events	Windows event logging
	Instrument firmware: • Error logs in `%PROGRAMDATA%\Rohde-Schwarz\Vna\Log` • Reports created on demand in `%PUBLIC%\Documents\Rohde-Schwarz\Vna\Reports`. Reports comprise memory dumps.	Instrument firmware
Environmental conditions	Cal unit characterizations: Comprise temperature info at creation time. Visible in the Characterize Cal Unit dialog and in persisted `*.calkit` files.	Instrument firmware (via cal unit interaction)
Usage and user behavior	Sweep counters (HUMS only) • Generic sweep counter • Sweep counters for certain K-Options Accessible via HUMS	HUMS

Collecting data

You can access, retrieve and delete most of the generated operational and non-operational data using standard Windows file and event management.

File locations are described in Section 10.2, "Storage locations", on page 1695, and Table 10-1. Events for the started Rohde & Schwarz ("Rs...") services can be filtered and extracted using the Windows event viewer.

Furthermore, the firmware can collect global instrument settings, instrument status information and certain non-operational data in a zip archive containing files in various text and binary formats ("Create R&S Support Information"). See Section 9.3, "Collecting information for technical support", on page 1691.

The volume of the non-operational data depends on the usage of the instrument. For each firmware start or error, the firmware adds a log entry of a few kilobytes.The size of a "R&S Support Information" zip file is typically several tens of megabytes.

To learn how to access the information collected via the Health and utilization monitoring service (HUMS), see the corresponding user manual on the Rohde & Schwarz internet.

Protecting data

For additional information on the different memory types of your product and on how to protect sensitive data, see the instrument security manual that is available on the Rohde & Schwarz internet.

10.2 Storage locations

This section describes the most important storage locations of the firmware application and its files.

Use standard Windows tools and techniques to manage files on the instruments hard drive and/or connected network drives. To a certain extent, file management can also be done via SCPI (see Section 7.3.11, "MMEMory commands", on page 1254).

10.2.1 Firmware application files

The instrument firmware is installed in the directory

`%PROGRAMFILES%\Rohde-Schwarz\Vector Network Analyzer\ZNB`,

where `%PROGRAMFILES% = C:\Program Files` in delivery state.

10.2.2 Firmware-generated operational data

The data representing the current operational state of the instrument is available at

`%PROGRAMDATA%\Rohde-Schwarz\Vna`,

where `%PROGRAMDATA% = C:\ProgramData` in delivery state.

It is maintained by the instrument firmware and contains:

- Global (persistent) settings and the currently loaded Section 4.1.2, "Recall sets", on page 80 (in subfolder `RestartSettings`)
- Log files (in subfolder `Logs`)
- Other static and runtime resources

- The content of the `%PROGRAMDATA%\Rohde-Schwarz\Vna` and other operational and non-operational data can be exported using Create R&S Support Information. The resulting zip archive is saved to the subfolder `Reports` of the shared user data directory `%PUBLIC%\Documents\Rohde-Schwarz\Vna`.
- Close the instrument firmware and use the "Reset Factory State" script to reset `%PROGRAMDATA%\Rohde-Schwarz\Vna` (i.e. the instrument) to its factory default. The script can be accessed from the instrument's program group in the Windows Start menu.
- The "Reset Factory State" does not reset or delete the user-generated operational data.

10.2.3 User-generated operational data

When storing user-generated operational data (saved recall sets, trace files, de-/embedding files, etc.) via GUI or SCPI, by default the firmware selects a target directory within the shared documents folder

`%PUBLIC%\Documents\Rohde-Schwarz\Vna`.

In delivery state, `%PUBLIC% = C:\Users\Public`.

A factory reset of the instrument ("Reset Factory State" script) does not delete any user-generated operational data.

11 Transporting

Lifting and carrying

See:

- "Lifting and carrying the product" on page 16
- Section 3.1.1, "Lifting and carrying", on page 21.

Packing

Use the original packaging material. It consists of antistatic wrap for electrostatic protection and packing material designed for the product.

If you do not have the original packaging, use similar materials that provide the same level of protection. You can also contact your local Rohde & Schwarz service center for advice.

Securing

When moving the product in a vehicle or using transporting equipment, make sure that the product is properly secured. Only use items intended for securing objects.

Transport altitude

Unless otherwise specified in the data sheet, the maximum transport altitude without pressure compensation is 4500 m above sea level.

12 Maintenance, storage and disposal

We advise to check the nominal data from time to time.

12.1 Replacing fuses

For the R&S ZNB, each of the optional Bias tees at the rear panel (labeled PORT BIAS <no>) is protected by a fuse (0.250 A, F IEC60127-2/2, stock no. 0020.7246.00).

To replace the PORT BIAS input fuses (R&S ZNB only)

► Open the fuse holder by slightly turning the lid counterclockwise.

12.2 Recalibration and repair

For accurate measurements, the product must be recalibrated by Rohde & Schwarz service after the calibration interval in the data sheet has elapsed.

12.3 Cleaning

How to clean the product is described in "Cleaning the product" on page 17.

Do not use any liquids for cleaning. Cleaning agents, solvents, acids and bases can damage the front panel labeling, plastic parts and display.

12.4 Storage

Protect the product against dust. Ensure that the environmental conditions, e.g. temperature range and climatic load, meet the values specified in the specifications document.

12.5 Disposal

Rohde & Schwarz is committed to making careful, ecologically sound use of natural resources and minimizing the environmental footprint of our products. Help us by disposing of waste in a way that causes minimum environmental impact.

Disposing of electrical and electronic equipment

A product that is labeled as follows cannot be disposed of in normal household waste after it has come to the end of its life. Even disposal via the municipal collection points for waste electrical and electronic equipment is not permitted.



Figure 12-1: Labeling in line with EU directive WEEE

Rohde & Schwarz has developed a disposal concept for the eco-friendly disposal or recycling of waste material. As a manufacturer, Rohde & Schwarz completely fulfills its obligation to take back and dispose of electrical and electronic waste. Contact your local service representative to dispose of the product.

13 Annexes

The following sections cover mostly hardware and service-related topics.

13.1 Firmware update

Admin account

You need administrator rights to install a new firmware. See Section 3.1.9, "Windows operating system", on page 27 for details.

64-bit only

For analyzer firmware versions ≥ 3.0, 32-bit Windows is no longer supported. Only 64-bit firmware is available.

To upgrade your instrument to 64-bit Windows 10, contact Rohde & Schwarz service.

13.1.1 Update information

Rohde & Schwarz firmware releases are thoroughly tested concerning backward compatibility of features and performance.

A prerequisite for publishing a new firmware release is the fact that according to our test results a re-calibration is not required after updating an instrument to this new version.

If a firmware release does not fulfill this prerequisite, a corresponding note is added to the Release Notes.

13.1.2 Firmware downgrade

Generally, it is not recommended to use an earlier version than the latest version available. In some cases, the older versions do not support the hardware used in your instrument.

Usually you can downgrade to a previous version by executing the related firmware setup file.

Downgrade to an earlier firmware version

To downgrade the firmware from a version ≥ 4.00 to an earlier version, it is required to uninstall the firmware using the **uninstall shortcut** on the Windows desktop before proceeding with the installation.

13.1.3 Firmware installation

The analyzer firmware is supplied as a single executable setup file (`*.exe`).

To install the firmware:

1. Copy the setup file to any storage medium accessible from the analyzer. This can be either the internal mass storage drive, an external storage medium (USB flash drive, external CD-ROM drive) or a network drive (LAN).
   The default name of the internal drive is `C:`. External storage devices are automatically mapped to the next free drive, i.e. `D:`, `E:` etc.
2. Run the setup file from the Windows® Explorer. Follow the instructions of the setup wizard.
   Setup files can be reinstalled.

13.2 System recovery

The system drive of the R&S ZNB is delivered with a recovery partition that allows you to restore the original operating system image and firmware.

To access the recovery functionality, select "Settings" > "Update & Security" from the Windows® Start menu. Click "Restart Now" and wait for the "R&S Recovery Environment" to start.



You can also use the Advanced tab of the "System Config" dialog to boot into the recovery environment.

To restore the original operating system image and firmware, proceed as follows:

1. In the "R&S Recovery Environment" select "Factory Default Restore"
2. Wait for the reimaging process to complete
3. Reboot the instrument

After the restore, upgrade to the desired firmware version (see Section 13.1, "Firmware update", on page 1700).

13.3 Interfaces and connectors

This chapter provides a detailed description of the rear panel connectors of the R&S ZNB. An overview of the available front and rear panel is given in the Getting Started guide (see Section 3.2, "Instrument tour", on page 35).

EMI Suppression
Notice the instructions in "Cable selection and electromagnetic interference (EMI)" on page 24.

13.3.1 Rear panel connectors

The rear panel of the R&S ZNB provides various connectors for external devices and control signals.

13.3.1.1 USER PORT

25-pole D-Sub connector used as an input and output for low-voltage (3.3 V) TTL control signals. Some of the lines can be configured (see CONTrol commands and OUTPut commands).



Table 13-1: VNA User Port: Pole Assignment

Pin No.	Name	Input (I) or Output (O)	Function
1	AGND	-	Ground
2	UC_EXT_TRG_IN	I	External trigger 1 input, 5 V tolerant *)
3	AGND	-	Ground
4	UC_BUSY	O	Hardware measurement time
5	AGND	-	Ground
6	READY FOR TRIGGER	O	Measurement completed, ready for new trigger
7	AGND	-	Ground
8	UC_CH_BIT0	O	Channel bit 0; see `CONTrol:AUXiliary:C[:DATA]` and `OUTPut<Ch>:UPORt[:VALue]`
9	UC_CH_BIT1	O	Channel bit 1
10	UC_CH_BIT2	O	Channel bit 2
11	UC_CH_BIT3	O	Channel bit 3
12	AGND	-	Ground
13	UC_PASS1	O	Pass/fail result of limit check 1; TTL Out Pass 1 (see TTL1 Pass / TTL2 Pass)
14	UC_PASS2	O	Pass/fail result of limit check 2; TTL Out Pass 2
15	AGND	-	Ground
16	UC_DRV_PORT1	O	Used as drive ports (`OUTPut:UPORt:ECBits OFF`) or channel bits (`OUTPut:UPORt:ECBits ON`; default)

Pin No.	Name	Input (I) or Output (O)	Function
17	UC_DRV_PORT2	O	If used as drive ports, DRIVE PORT *i* is active while test port *i* is the source port. If used as channel bits, the pin states can be defined using `OUTPut<Ch>:UPORt[:VALue]`.
18	UC_DRV_PORT3	O	
19	UC_DRV_PORT4	O	
20	AGND	-	Ground
21	UC_EXT_GEN_TRG	O	Control signal for external generator
22	UC_EXT_GEN_BLANK	I	Handshake signal from external generator
23	AGND	-	Ground
24	UC_FOSW	I	Control input A, 5 V tolerant
25	UC_TRG2	I	External trigger 2 input, 5 V tolerant

*) Feeding in the external trigger signal via the BNC connector Trigger In is equivalent. The minimum pulse width of the trigger signals is 1 µs.

EMI Suppression

Use only double shielded cables or disconnect the input pins of the USER PORT connector to avoid spurious input signals which may cause undesirable events.

This is of particular importance for the external trigger input (pin no. 2) if the Trigger In input is used.

13.3.1.2 Trigger In



BNC female connector for external trigger input; see "Free Run (None) / External / Manual / Multiple" on page 553. The external trigger input signal must be a 3.3 V or 5 V TTL signal with a minimum pulse width of 1 µs. The trigger input has a high input impedance (> 10 kΩ).

13.3.1.3 Trigger Out



BNC female connectors for external trigger output; see Section 5.10.4, "Trigger Out tab", on page 557.

The R&S ZNB trigger output provides a configurable 5 V TTL trigger signal. The output impedance is approximately 50 Ω.

13.3.2 LAN interface

To be integrated in a LAN, the instrument is equipped with a LAN interface, consisting of an RJ-45 connector, a network interface card and protocols. The network interface card supports IEEE 802.3 for a 10 Mbps Ethernet and IEEE 802.3u for a 100 Mbps Ethernet.

Instrument access is possible via the VXI-11 protocol. It is usually achieved from high level programming platforms by using the Virtual Instrument Software Architecture (VISA) library as an intermediate abstraction layer. VISA encapsulates the low-level function calls and thus makes the transport interface transparent for the user. See also Section 6.1, "Introduction to remote control", on page 868.

13.3.2.1 VXI-11 interface messages

The following VXI-11 interface messages (also termed low-level control messages) are also supported by the GPIB Explorer.

Command	Meaning	Effect on the instrument
@DCL	Device Clear	Aborts the processing of the commands just received and sets the command processing software to a defined initial state. Does not change the instrument settings.
@GET	Group Execute Trigger	Triggers the active measurement sequence (e.g. a sweep). The effect of the command is the same as with that of a pulse at the external trigger signal input.
@LOC	Go to Local	Transition to the "local" state (manual control).
@REM	Go to Remote	Transition to the "remote" state (remote control).

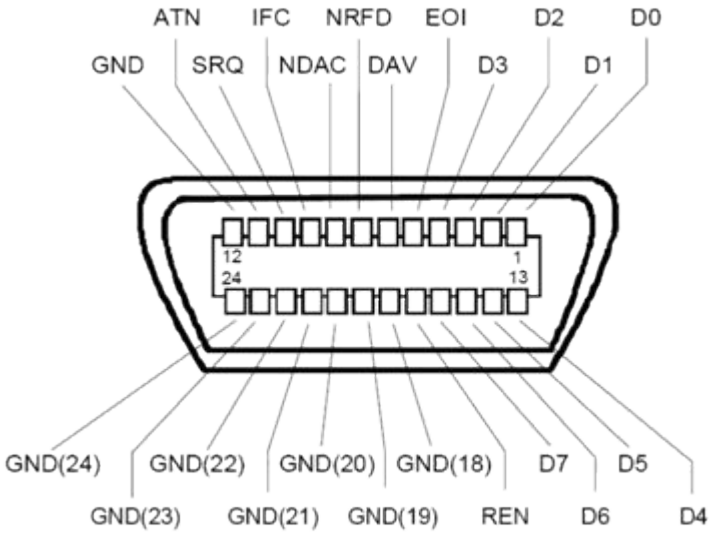
13.3.3 GPIB interface

The R&S ZNB can be equipped with a GPIB (IEC/IEEE) bus interface (option R&S ZNB-B10. The interface connector labeled "GPIB" is located on the rear panel of the instrument. The GPIB bus interface is intended for remote control of the R&S ZNB from a controller.

Characteristics of the interface

- 8-bit parallel data transfer
- Bidirectional data transfer
- Three-line handshake
- High data transfer rate of max. 1 MByte/s
- Up to 15 devices can be connected
- Wired OR if several instruments are connected in parallel

Pin assignment



Bus lines

- Data bus with 8 lines D0 to D7:
  The transmission is bit-parallel and byte-serial in the ASCII/ISO code. D0 is the least significant bit, D7 the most significant bit.
- Control bus with five lines:
  IFC (Interface Clear): active LOW resets the interfaces of the instruments connected to the default setting.
  ATN (Attention): active LOW signals the transmission of interface messages, inactive HIGH signals the transmission of device messages.
  SRQ (Service Request): active LOW enables the connected device to send a service request to the controller.
  REN (Remote Enable): active LOW permits switchover to remote control.
  EOI (End or Identify): has two functions in connection with ATN:
  - ATN=HIGH active LOW marks the end of data transmission.
  - ATN=LOW active LOW triggers a parallel poll.
- Handshake bus with three lines:
  DAV (Data Valid): active LOW signals a valid data byte on the data bus.
  NRFD (Not Ready For Data): active LOW signals that one of the connected devices is not ready for data transfer.
  NDAC (Not Data Accepted): active LOW signals that the instrument connected is accepting the data on the data bus.

The R&S ZNB provides several functions to communicate via GPIB bus. They are described in the following sections.

13.3.3.1 Interface functions

Instruments which can be controlled via GPIB bus can be equipped with different interface functions. The interface functions for the R&S ZNB are listed in the following table.

Control character	Interface function
SH1	Handshake source function (source handshake), full capability
AH1	Handshake sink function (acceptor handshake), full capability
L4	Listener function, full capability, de-addressed by MTA.
T6	Talker function, full capability, ability to respond to serial poll, deaddressed by MLA
SR1	Service request function (Service Request), full capability
PP1	Parallel poll function, full capability
RL1	Remote/Local switch over function, full capability
DC1	Reset function (Device Clear), full capability
DT1	Trigger function (Device Trigger), full capability

13.3.3.2 Interface messages

Interface messages are transmitted to the instrument on the data lines, with the attention line being active (LOW). They serve to communicate between controller and instrument.

Universal commands

Universal commands are encoded in the range 10 through 1F hex. They are effective for all instruments connected to the bus without previous addressing.

Command	QuickBASIC command	Effect on the instrument
DCL (Device Clear)	IBCMD (controller %, CHR$(20))	Aborts the processing of the commands just received and sets the command processing software to a defined initial state. Does not change the instrument settings.
IFC (Interface Clear)	IBSIC (controller%)	Resets the interfaces to the default setting.
LLO (Local Lockout)	IBCMD (controller %, CHR$(17))	The LOC/IEC ADDR key is disabled.
SPE (Serial Poll Enable)	IBCMD (controller %, CHR$(24))	Ready for serial poll

Command	QuickBASIC command	Effect on the instrument
SPD (Serial Poll Disable)	IBCMD (controller %, CHR$(25))	End of serial poll
PPU (Parallel Poll Unconfigure)	IBCMD (controller %, CHR$(21))	End of the parallel-poll state

Addressed commands

Addressed commands are encoded in the range 00 through 0F hex. They are only effective for instruments addressed as listeners.

Command	QuickBASIC command	Effect on the instrument
GET (Group Execute Trigger)	IBTRG (device%)	Triggers a previously active device function (e.g. a sweep). The effect of the command is the same as with that of a pulse at the external trigger signal input.
GTL (Go to Local)	IBLOC (device%)	Transition to the "Local" state (manual control)
PPC (Parallel Poll Configure)	IBPPC (device%, data%)	Configures the instrument for parallel poll. Additionally, the QuickBASIC command executes PPE/PPD.
SDC (Selected Device Clear)	IBCLR (device%)	Aborts the processing of the commands just received and sets the command processing software to a defined initial state. Does not change the instrument setting.

13.3.3.3 Instrument messages

Instrument messages (commands) are transferred on the data lines of the GPIB bus while the ATN line is not active. ASCII code is used.

Structure and syntax of the instrument messages are described in Section 7, "Command reference", on page 911. The chapter also provides a detailed description of all messages implemented by the analyzer.

13.3.4 Handler I/O (universal interface)

Option R&S ZN-B14 / R&S ZNBT-Z14

A network analyzer which is equipped with a Handler I/O (universal interface) option, can interact with an external part handler. The digital control signals on the interface connector indicate the possible start and the end of a measurement, and a global limit check result. Typically, the handler inserts the device to be tested into a test fixture, provides a trigger pulse to initiate the measurement, removes and replaces the device

after the measurement is complete, and sort it into pass/fail bins. A sample flow diagram for this process is shown below.

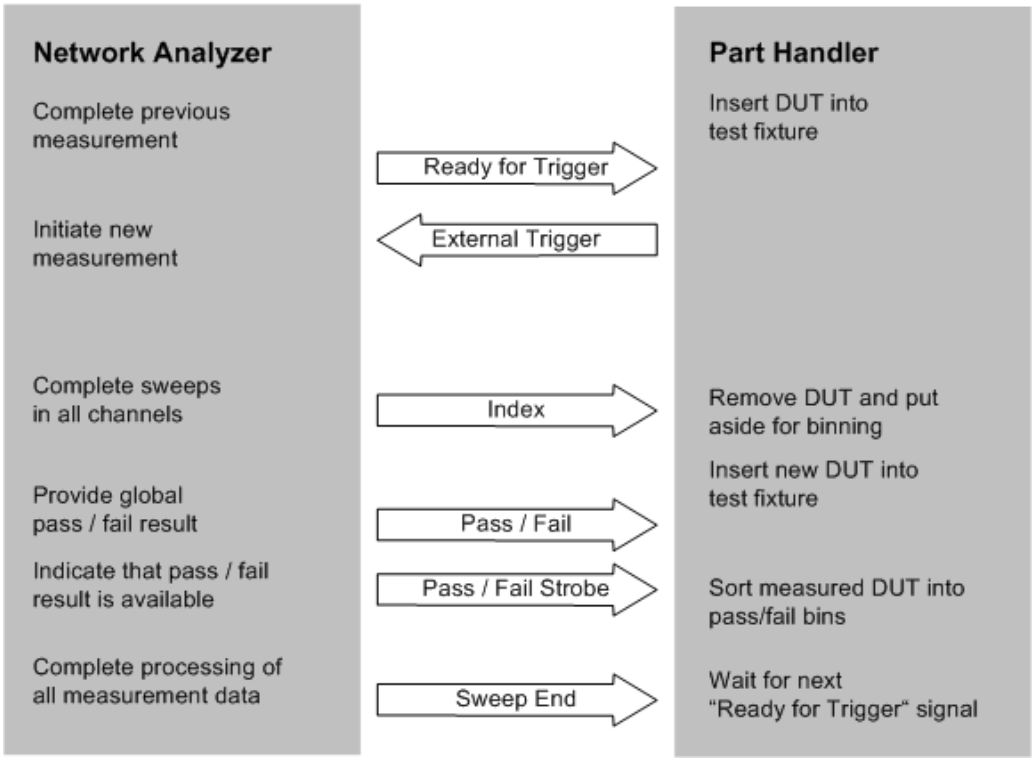


Figure 13-1: Possible stages of an automated test

Preparation of the network analyzer and the part handler

The network analyzer configuration depends on the measurement to be made. Starting from the preset state, usually you have to adjust the following settings:

1. Enable external trigger:
   Channel – [Trigger] > "Trigger" > "External"
2. (Optional:) Select single sweep mode: Channel – [Sweep] > "Sweep Control" > "Single"
3. Define limit lines and enable the limit check: Trace – [Line] > "Limit Test" > ...

The Universal Interface connector must be connected to the part handler using an appropriate cable. If necessary, configure the data ports to ensure that network analyzer and part handler are compatible.

13.3.4.1 Control signals

Most of the signals in the figure below are controlled by the measurement. It is possible though to query the transitions of the "Input 1" signal, to configure the "Output 1", "Output 2", "Sweep End", and "Pass/Fail" signals and to route the "Index" and "Ready for Trigger" signals using SCPI commands.

- `CONTrol:HANDler:INPut?` queries the high to low transitions of the "Input 1" signal.
- The `CONTrol:HANDler:OUTPut<Pt>...` commands set the output signals to a definite state and specify whether this state will change to "Low" when the Input 1 signal goes to "Low". This mechanism provides either static output signals or output signals which are controlled by Input 1.
- The "Index" and "Ready for Trigger" signals can be routed to pins 20 and 21 of the Universal Interface connector, where they replace the PORT B6 and PORT B7 input/output signals. See Section 13.3.4.2, "Data ports", on page 1709.
- The `CONTrol:HANDler:PASSfail...` commands configure the "Pass/Fail" signal and query the global pass/fail status of the last measurement.

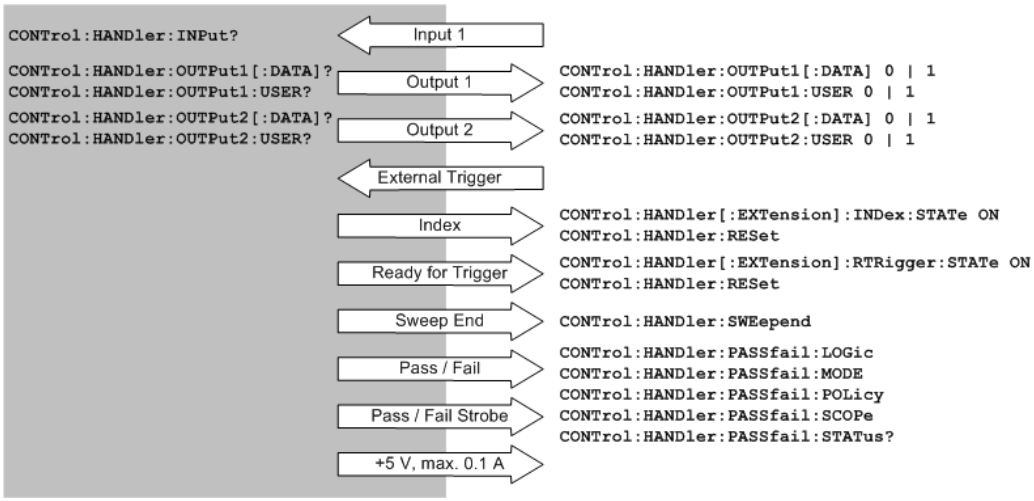


Figure 13-2: Control signals and power supply

SCPI commands: See Section 7.3.3, "CONTrol commands", on page 1183

13.3.4.2 Data ports

In addition to the control signals, the Universal Interface provides four bi-directional data ports A, B, C, D. The data ports must be configured explicitly using SCPI commands; they are not controlled by the measurement.

- With an output data port, you can configure the part handler or other devices used in testing from the network analyzer.
- With an input data port, you can configure the network analyzer using external signals and an appropriate control program.

Device configurations via data port signals are usually performed in a preliminary stage, before the actual measurement sequence. If the "Index" and "Ready for Trigger" signals are enabled at this stage (see Section 13.3.4.1, "Control signals", on page 1708), they replace the PORT B6 and PORT B7 signals. Port B can still be used as a 6-bit parallel input/output port.

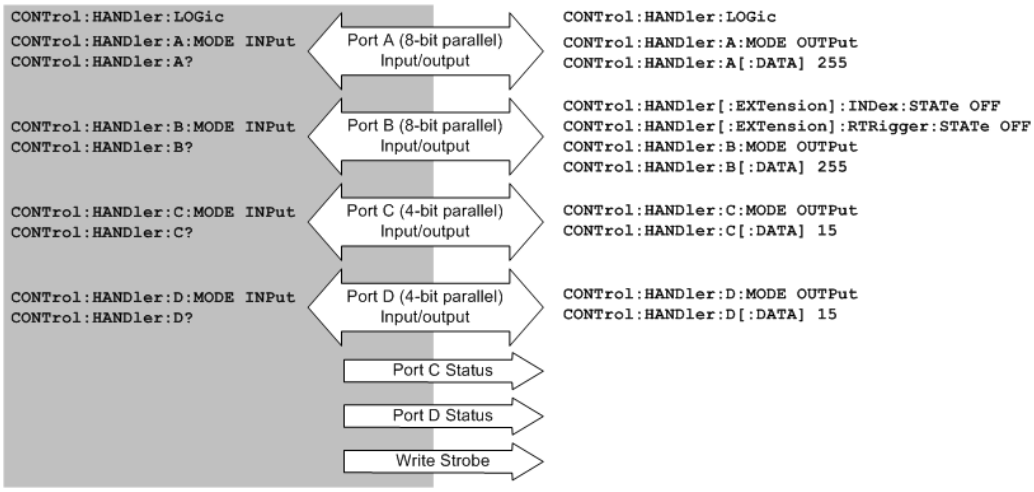


Figure 13-3: Data ports and related signals

Combined ports (ports E, F, G, H)

A combination of ports A / B and ports C /D provides two additional 16-bit and 8-bit-wide bidirectional ports. The combined ports are termed ports E and F, respectively. A combination of ports A / B / C and ports A / B / C / D provides two 20-bit and 24-bit-wide bidirectional ports. These combined ports are termed ports G and H.

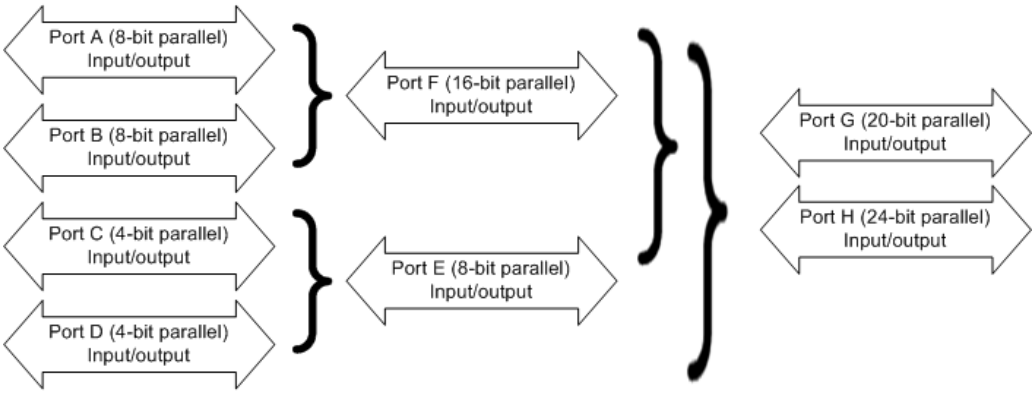


Figure 13-4: Definition of ports E, F, G, H

The properties of the combined ports are as follows:

- The signal direction (input or output) is according to the configuration of ports A, B, C, and D. E.g. to write data to port G (H), ports A, B, and C (A, B, C, and D) must be configured as output ports.
- Data can be read and written using the `CONTrol:HANDler:E|F|G|H[:DATA]` commands.
- The bit order is D3 ... D0 C3 ... C0 (port E), B7 ... B0 A7 ... A0 (port F), C3 ... C0 B7 ... B0 A7 ... A0 (port G), and D3 ... D0 C3 ... C0 B7 ... B0 A7 ... A0 (port H).

SCPI commands: See Section 7.3.3, "CONTrol commands", on page 1183

13.3.4.3 Universal interface connector

The Handler I/O (Universal Interface) option includes a Centronics 36 input/output connector.

The R&S ZNB's internal Handler I/O option R&S ZN-B14 is placed in the right-hand part of the network analyzer's rear panel. It must be installed by a Rohde & Schwarz service representative.

The pin assignment of the connector is shown below. A slash (/) at the beginning of the signal name indicates that, by default, it is an active low (negative logic) signal.

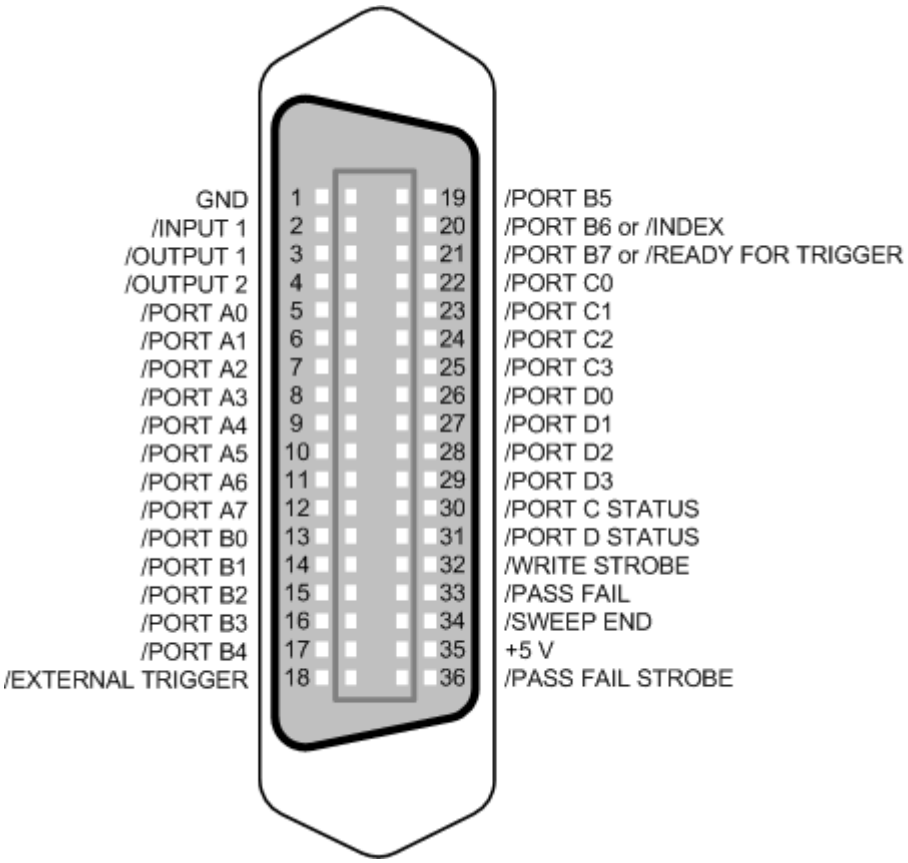


Figure 13-5: Pin assignment of the Universal Interface connector

The input and output signals at the connector are described below.

Pin No.	Signal	Input (I) or Output (O)	Description
1	GND	–	Ground
2	/INPUT 1	I	Configurable input/output logic.

Pin No.	Signal	Input (I) or Output (O)	Description
3	/OUTPUT 1	O	When a negative pulse is fed to /INPUT 1, then /OUTPUT <Pt> is switched to "low" ("high"), if CONTrol:HANDler:OUTPut<Pt>:USER is set to 1 (0). The current state of /OUTPUT <Pt> can be set using CONTrol:HANDler:OUTPut<Pt>[:DATA].
4	/OUTPUT 2	O	
5	/PORT A0	I or O	Port A, bit no. 0 (8-bit parallel input or output port)
6	/PORT A1	I or O	Port A, bit no. 1
7	/PORT A2	I or O	Port A, bit no. 2
8	/PORT A3	I or O	Port A, bit no. 3
9	/PORT A4	I or O	Port A, bit no. 4
10	/PORT A5	I or O	Port A, bit no. 5
11	/PORT A6	I or O	Port A, bit no. 6
12	/PORT A7	I or O	Port A, bit no. 7
13	/PORT B0	I or O	Port B, bit no. 0 (8-bit parallel input or output port)
14	/PORT B1	I or O	Port B, bit no. 1
15	/PORT B2	I or O	Port B, bit no. 2
16	/PORT B3	I or O	Port B, bit no. 3
17	/PORT B4	I or O	Port B, bit no. 4
18	/EXTERNAL TRIGGER	I	External trigger signal
19	/PORT B5	I or O	Port B, bit no. 5
20	/PORT B6 or /INDEX	I or O O	Port B, bit no. 6 The /INDEX signal changes to "Low" when a measurement is complete (all sweeps in all channels have been performed; the DUT can be removed, but possibly the measurement results are not valid yet).
21	/PORT B7 or /READY FOR TRIGGER	I or O O	Port B, bit no. 7 The /READY FOR TRIGGER signal changes to "Low" when the analyzer is ready to receive a trigger for a new measurement.
22	/PORT C0	I or O	Port C, bit no. 0 (4-bit parallel input or output port)
23	/PORT C1	I or O	Port C, bit no. 1
24	/PORT C2	I or O	Port C, bit no. 2
25	/PORT C3	I or O	Port C, bit no. 3
26	/PORT D0	I or O	Port D, bit no. 0 (4-bit parallel input or output port)
27	/PORT D1	I or O	Port D, bit no. 1
28	/PORT D2	I or O	Port D, bit no. 2
29	/PORT D3	I or O	Port D, bit no. 3

Pin No.	Signal	Input (I) or Output (O)	Description
30	/PORT C STATUS	O	Indicates the status of port C: • Low: port C is an input port • High: port C is an output port
31	/PORT D STATUS	O	Indicates the status of port D: • Low: port D is an input port • High: port D is an output port
32	/WRITE STROBE	O	Changes to "Low" when valid data is present at any of the output ports (i.e. when the output level at any port changes).
33	/PASS FAIL	O	Indicates the status of the global limit check for the last measurement: • Low: global limit check failed • High: global limit check passed Other configurations are possible; see `CONTrol:HANDler:PASSfail...` commands.
34	/SWEEP END	O	Indicates the end of the measurement. Changes to "Low" after all sweeps in all channels have been completed and the measurement data has been processed. Alternative configurations: see `CONTrol:HANDler:SWEepend`.
35	+5 V DC	O	DC power supply for external devices; +5 V ± 250 mV, max. 0.1 A
36	/PASS FAIL STROBE	O	Changes to "Low" when limit check results are present on /PASS FAIL (pin no. 33).

All digital input signals must be TTL compatible with an allowed input voltage range between –0.5 V and +5.5 V. The circuit diagram of the input path is shown below.

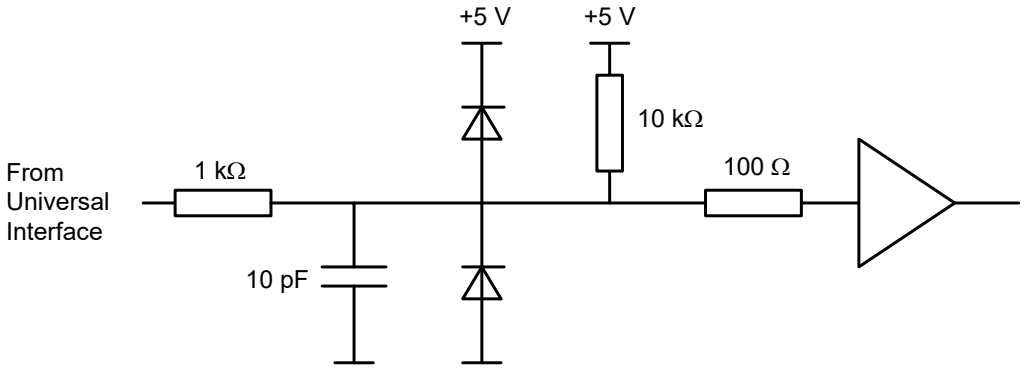


Figure 13-6: Circuit diagram of the input path

Digital output signals are low-voltage TTL compatible with output voltages between 0 V and +5 V. The circuit diagram of the output path is shown below.

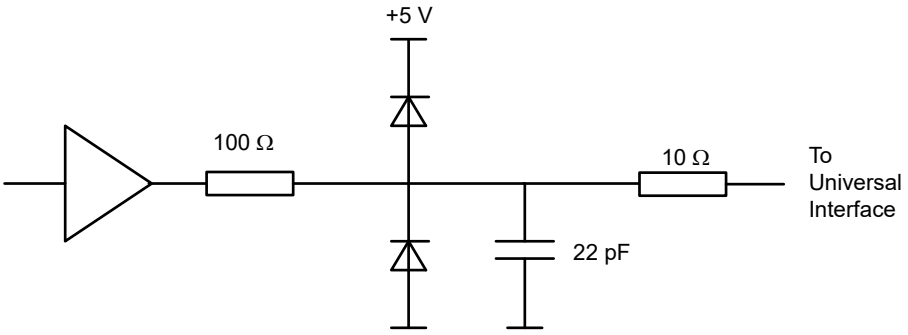


Figure 13-7: Circuit diagram of the output path

The default configuration of the signals is listed below. Notice that `*RST` or "Preset" do not change the configuration of the Universal Interface. Use `CONTrol:HANDler:RESet` to restore default values.

Signal	*RST Configuration
Port A, Port B	Output ports, all bits "High" (decimal 0)
Port C, Port D	Input ports
/PORT C STATUS, /PORT D STATUS	"Low"
/OUTPUT 1, /OUTPUT 2	"High"
/SWEEP END	"High"
/PASS_FAIL	"High"

SCPI commands: `CONTrol:HANDler:RESet` (resets all configurable signals)

13.3.4.4 Timing of control signals

The timing of the essential measurement control signals is shown in the figure below. The duration of the shaded time intervals depends on the measurement settings.

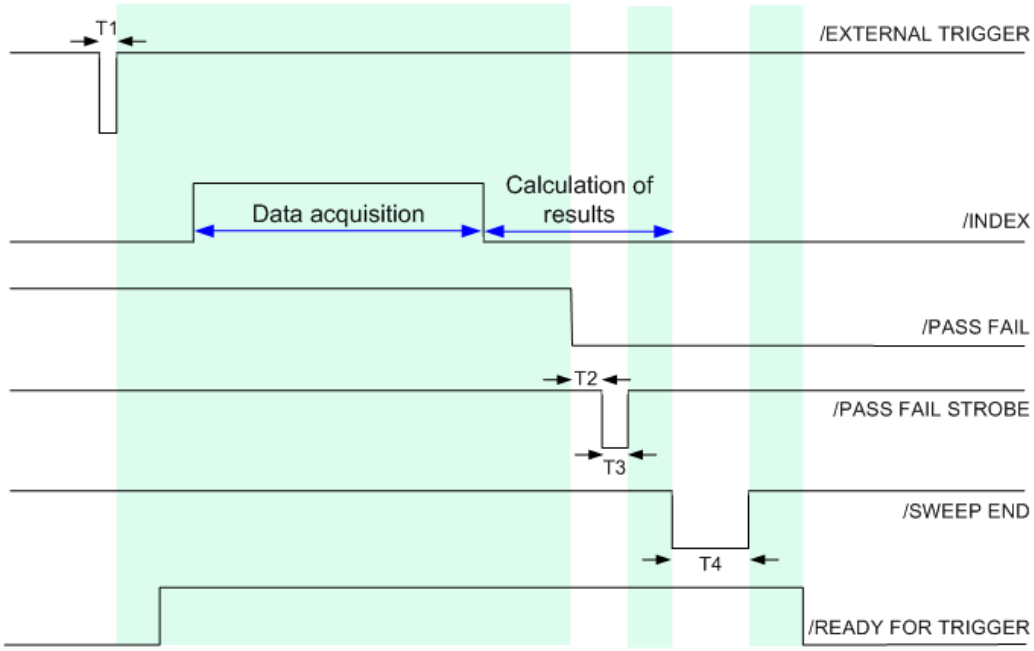


Figure 13-8: Timing of measurement control signals

The figure above corresponds to the default configuration of the /PASS FAIL signal. For alternative configurations, refer to the documentation of the `CONTrol:HANDler:PASSfail...` commands in Section 7.3.3, "CONTrol commands", on page 1183. The figure contains the following pulse durations and response times.

Time	Description	Value
T1	Pulse duration of /EXTERNAL TRIGGER	Minimum value: 1 µs
T2	Response time of /PASS FAIL STROBE	1 µs
T3	Pulse duration of /PASS FAIL STROBE	1 µs
T4	Pulse duration of /SWEEP END	12 µs

The timing of the data port signals and the input/output signals is as follows:

- The low pulse of the /WRITE STROBE signal occurs approx. 0.1 µs after a value is written to the output ports A to D. The pulse duration of the /WRITE STROBE signal is 1 µs.
- The low pulse of the /OUTPUT 1 or /OUTPUT 2 signals (if enabled) occur approx. 0.6 µs after the falling edge of the /INPUT 1 signal.

13.3.5 RFFE GPIO interface

For the R&S ZNB, an optional RFFE GPIO interface board is available.

13.3.5.1 Pin assignment

This extension board is equipped with a standard 25-pin female D-sub connector providing 2 independent RF Front-End (RFFE) interfaces according to the MIPI® Alliance "System Power Management Interface Specification" and 10 General Purpose Input/ Output (GPIO) ports.

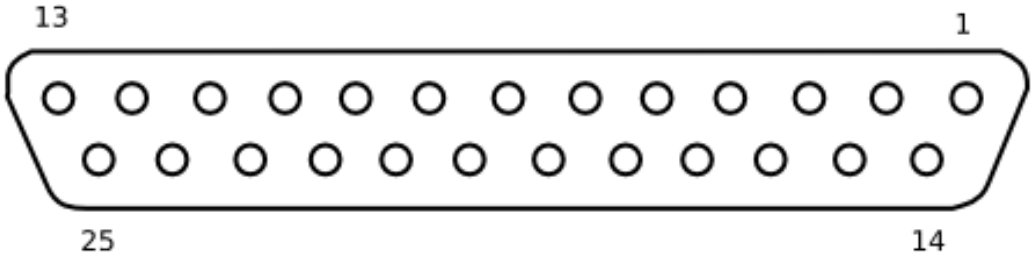


Figure 13-9: D-sub 25-pin female connector, front view

Table 13-2: PIN mapping RFFE - GPIO extension board connector

PIN number	Comment
1,3,5,11,22	Ground
2	RFFE1_VIO
4	RFFE2_VIO
6	GPIO 2
7	GPIO 4
8	GPIO 6
9	GPIO 8
10	GPIO 9
12	For future use, please do not connect
13	For future use, please do not connect
14	RFFE1_CLK
15	RFFE1_DATA
16	RFFE2_CLK
17	RFFE2_DATA
18	GPIO 1
19	GPIO 3
20	GPIO 5
21	GPIO 7
23	GPIO 10

PIN number	Comment
24	For future use, please do not connect
25	For future use, please do not connect

13.3.5.2 Interface description

The values in the table below are typical values. See the R&S ZNB data sheet or the R&S ZN-Z15 data sheet for details.

Table 13-3: RFFE bus interface

Parameter	MIN [V]	MAX [V]	Step size [V]
IO voltage	0	2.5	0.001
Low voltage	0	2.5	0.001
High voltage	0	2.5	0.001
		MAX [mA]	
Current		20	
	MIN [kHz]	**MAX [kHz]**	**Possible values [kHz]**
Clock frequency	31.25	26000	52000/n with n=1664, ..., 2

All remaining data (e.g. rise time) are according to the specification v.1.00 of the MIPI Alliance Group.

Table 13-4: GPIO Interface

Parameter	MIN [V]	MAX [V]	Step size [V]
	-7	+15	0.005
		MAX [mA]	
Current GPIO 1, ..., 8		20	
Current GPIO 9, 10		100	

The output voltages of the RFFE and GPIO signals do not have an offset to compensate additional losses into account. Please adjust the voltage level directly on your board or at the pins of the connected device.

Table 13-5: Voltage/Current Measurement (Variant 03 only)

	Range	Resolution
Voltage		

RFFE 1 and 2 (VIO/DATA/CLK)	0 V to +3 V	100 µV
GPIO 1 to 10	-5 V to +10 V	100 µV
Current*		
RFFE 1 and 2 (VIO/DATA/CLK), GPIO 1 to 8		
10 Ω source resistance (shunt)	–20 mA to +20 mA	10 µA
100 Ω source resistance (shunt)	–2 mA to +2 mA	1 µA
1 kΩ source resistance (shunt)	–200 µA to +200 µA	100 nA
10 kΩ source resistance (shunt)	–20 µA to +20 µA	10 nA
100 kΩ source resistance (shunt)	–2 µA to +2 µA	1 nA
GPIO 9 and 10		
	–100 mA to +100 mA	10 µA
* the current values are valid if the GPIO voltages are within -5 V to +9 V		

13.4 Showroom mode

In "Showroom Mode" the R&S ZNB uses a configurable recall set whenever the device is (re-)started or the [Preset] key is pressed. However, it can only be enabled by directly editing the Windows registry.

1. Run the R&S ZNB and configure it as required for the intended showroom operation.
2. Save the configuration as described in Section 5.14.1, "Recall sets tab", on page 779. If necessary, move the recall set file (`*.znx|*.znxml`) to the appropriate location.
3. Under the registry key `HKEY_LOCAL_MACHINE/SOFTWARE/Rohde-Schwarz/Vna`, create a new subkey `ShowroomMode`.
4. Within the `ShowroomMode`-subkey, create the string value `ShowroomSetup`.
5. Modify the value `ShowroomSetup`: set its value data to the full path of the recall set. Use forward slashes as path separators, e.g. "C:/Documents and Settings/All Users/Rohde-Schwarz/Vna/RecallSets/ShowroomSet1.znx".

   After the analyzer is restarted or "Preset", the specified recall set is loaded.

13.5 ENA emulation commands

The following table lists the commands implemented/overwritten for the ENA remote language.

ENA Emulation Command	R&S ZNB Default Parser Command
CALCulate:CORRection	no action, query always returns 1 (correction enabled)
CALCulate<Ch>:FSIMulator:SENDed:DEEMbed:PORT<Pt>[:TYPE] {USER | NONE}	CALCulate<Ch>:TRANsform:VNETworks:SENDed:DEEMbedding<PhyPt>:TNDefinition
CALCulate<Ch>:FSIMulator:SENDed:DEEMbed:PORT<Pt>:USER:FILename <string>	CALCulate<Ch>:TRANsform:VNETworks:SENDed:DEEMbedding<PhyPt>:PARameters:DATA
CALCulate<Ch>:FSIMulator:SENDed:DEEMbed:STATe {ON | OFF | 1 | 0}	n.a. (enabled channel-by-channel, port-by-port)
CALCulate{1..4}:FSIMulator:SENDed:PMCircuit:STAT {ON | OFF | 1 | 0}	CALCulate<Ch>:TRANsform:VNETworks:SENDed:EMBedding<PhyPt>[:STATe]
CALCulate{1..4}:FSIMulator:SENDed:PMCircuit:PORT<Pt>[:TYPE] {NONE | PCSC | PCSL | PLPC | PLSC | PLSL | SCPC | SCPL | SLPC | SLPL | USER}	CALCulate<Ch>:TRANsform:VNETworks:SENDed:EMBedding<PhyPt>:TNDefinition
CALCulate{1..4}:FSIMulator:SENDed:PMCircuit:PORT<i>:PARameter:C <numeric>	CALCulate<Ch>:TRANsform:VNETworks:SENDed:EMBedding<PhyPt>:PARameters:C<Cmp>
CALCulate{1..4}:FSIMulator:SENDed:PMCircuit:PORT<i>:PARameter:G <numeric>	CALCulate<Ch>:TRANsform:VNETworks:SENDed:EMBedding<PhyPt>:PARameters:G<Cmp>
CALCulate{1..4}:FSIMulator:SENDed:PMCircuit:PORT<i>:PARameter:L <numeric>	CALCulate<Ch>:TRANsform:VNETworks:SENDed:EMBedding<PhyPt>:PARameters:L<Cmp>
CALCulate{1..4}:FSIMulator:SENDed:PMCircuit:PORT<i>:PARameter:R <numeric>	CALCulate<Ch>:TRANsform:VNETworks:SENDed:EMBedding<PhyPt>:PARameters:R<Cmp>
CALCulate{1..4}:FSIMulator:SENDed:ZCONversion:STATe {ON | OFF | 1 | 0}	[SENSe<Ch>:]LPORt<LogPt>:ZDEFault[:STATe]
CALCulate{1..4}:FSIMulator:SENDed:ZCONversion:PORT<i>:Z0[:R] <numeric>	[SENSe<Ch>:]PORT<PhyPt>:ZREFerence
CALCulate<Ch>:FSIMulator:STATe {ON | OFF | 1 | 0}	n.a. (enabled channel-by-channel, port-by-port)
CALCulate{1...4}[:SELected]:DATA:FDATa? CALCulate{1...4}[:SELected]:DATA:SDATa? CALCulate{1...4}[:SELected]:DATA:FMEMory?	CALCulate<Chn>:DATA
CALCulate<Ch>[:SELected]FORMat	CALCulate<Chn>:FORMat

ENA Emulation Command	R&S ZNB Default Parser Command
CALCulate{1..4}[:SELected]:FUNCtion:TYPE {PTPeak | STDEV | MEAN | MAXimum | MINimum}	CALCulate<Chn>:STATistics:RESult?
CALCulate{1..4}[:SELected]:FUNCtion:DOMain[:STATe] {ON | OFF | 1 | 0}	CALCulate<Chn>:STATistics:DOMain:USER
CALCulate{1..4}[:SELected]:FUNCtion:DOMain:STARt <value>	CALCulate<Chn>:STATistics:DOMain:USER:STARt
CALCulate{1..4}[:SELected]:FUNCtion:DOMain:STOP <value>	CALCulate<Chn>:STATistics:DOMain:USER:STOP
CALCulate{1..4}[:SELected]:FUNCtion:DATA?	CALCulate<Chn>:STATistics:RESult?
CALCulate{1..4}[:SELected]:FUNCtion:EXECute	None (no action)
CALCulate{1...4}[:SELected]:LIMit[:STATe] {ON | OFF | 1 | 0}	CALCulate<Chn>:LIMit:STATe
CALCulate{1...4}[:SELected]:LIMit:DISPlay[:STATe] {ON | OFF | 1 | 0}	CALCulate<Chn>:LIMit:DISPlay[:STATe]
CALCulate{1...4}[:SELected]:LIMit:FAIL?	CALCulate<Chn>:LIMit:FAIL?
CALCulate<Ch>[:SELected]:LIMit<Tr>:DATA	CALCulate<Chn>:LIMit:DATA
CALCulate{1...4}[:SELected]:MARKer{1...10}:COUPle {ON | OFF | 1 | 0}	CALCulate:MARKer:COUPled[:STATe]
CALCulate{1...4}[:SELected]:MARKer{1...10}:FUNCtion:TYPE {MAXimum| MINimum | PEAK | LPEak | RPEak | TARGet | LTARget | RTARget}	CALCulate<Chn>:MARKer<Mk>:FUNCtion[:SELect]
CALCulate{1...4}[:SELected]:MARKer{1...10}:DISCrete {ON | OFF | 1 | 0}	CALCulate<Chn>:MARKer<Mk>:MODE
CALCulate{1...4}[:SELected]:MARKer{1...10}:REFerence[:STATe] {ON | OFF | 1 | 0}	CALCulate<Chn>:MARKer<Mk>:REFerence[:STATe]
CALCulate{1...4}[:SELected]:MARKer{1...10}:FUNCtion:EXECute	CALCulate<Chn>:MARKer<Mk>:SEARch[:IMMediate]
CALCulate{1...4}[:SELected]:MARKer{1...10}[:STATe] {ON | OFF | 1 | 0}	CALCulate<Chn>:MARKer<Mk>[:STATe]
CALCulate{1...4}[:SELected]:MARKer{1...10}:X <numeric>	CALCulate<Chn>:MARKer<Mk>:X
CALCulate{1...4}[:SELected]:MARKer{1...10}:Y?	CALCulate<Chn>:MARKer<Mk>:Y

ENA Emulation Command	R&S ZNB Default Parser Command
CALCulate{1...4}[:SELected]:MATH: FUNCtion {NORMal | ADD | SUBTract | MULTiply | DIVide}	CALCulate<Chn>:MATH:FUNCtion
CALCulate{1...4}[:SELected]:MATH: MEMorize	CALCulate<Chn>:MATH:MEMorize
CALCulate{1...4}:PARameter:COUNt <numeric>	
CALCulate{1...4}:PARameter{1...7}: DEFine {S11 | S21 | S31| S41 | S12 | S22 | S32 | S42 | S13 | S23 | S33 | S43 | S14 | S24 | S34 | S44}	CALCulate<Ch>:PARameter:DEFine
CALCulate{1...4}:PARameter{1...7}: SELect [<string>]	CALCulate<Ch>:PARameter:SELect
CALCulate{1...4}[:SELected]:SMOothing: APERture <numeric>	CALCulate<Chn>:SMOothing:APERture
CALCulate{1...4}[:SELected]: SMOothing[:STATe] {ON | OFF | 1 | 0}	CALCulate<Chn>:SMOothing[:STATe]
CONTrol:HANDler:A[:DATA] <numeric>	CONTrol:HANDler:A[:DATA]
CONTrol:HANDler:B[:DATA] <numeric>	CONTrol:HANDler:B[:DATA]
CONTrol:HANDler:C[:DATA] <numeric>	CONTrol:HANDler:C[:DATA]
CONTrol:HANDler:D[:DATA] <numeric>	CONTrol:HANDler:D[:DATA]
CONTrol:HANDler:A:MODE {INPut | OUTPut}	CONTrol:HANDler:A:MODE
CONTrol:HANDler:B:MODE {INPut | OUTPut}	CONTrol:HANDler:B:MODE
CONTrol:HANDler:C:MODE {INPut | OUTPut}	CONTrol:HANDler:C:MODE
CONTrol:HANDler:D:MODE {INPut | OUTPut}	CONTrol:HANDler:D:MODE
CONTrol:HANDler[:EXTension]:INDex: STATe {ON | OFF | 1 | 0}	CONTrol:HANDler[:EXTension]:INDex: STATe
CONTrol:HANDler:OUTPut{1...2}[:DATA] {1 | 0}	CONTrol:HANDler:OUTPut<Pt>[:DATA]
DISPlay:ANNotation:MESSage:STATe {ON | OFF | 1 | 0}	SYSTem:ERRor:DISPlay:STATe
DISPlay:ARRange {TILE | CASCade | OVERlay | STACk | SPLit | QUAD}	DISPlay:LAYout CASCade not supported (ignored)
DISPlay:CCLear	*CLS is the closest match, but this will also clear the error queue.
DISPlay:ENABle {ON | OFF | 1 | 0}	SYSTem:DISPlay:UPDate
DISPlay:SPLit	DISPlay:LAYout:EXECute

ENA Emulation Command	R&S ZNB Default Parser Command
DISPlay:UPDate[:IMMediate]	SYSTem:DISPlay:UPDate
DISPlay:VISible {ON | OFF | 1 | 0}	SYSTem:DISPlay:UPDate
DISPlay:WINDow{1...4}:ACTivate <numeric_value>	INSTrument:NSELect
DISPlay:WINDow{1...4}:MAXimize {ON | OFF | 1 | 0}	DISPlay[:WINDow<Wnd>]:MAXimize
DISPlay:WINDow{1...4}:SIZE {NORMal | MAXimized | MINimized}	DISPlay[:WINDow<Wnd>]:MAXimize MINimized mapped to NORMal
DISPlay:WINDow:SPLit	
DISPlay:WINDow{1...4}:TITLe:DATA <string>	DISPlay[:WINDow<Wnd>]:TITLe:DATA
DISPlay:WINDow{1...4}:TITLe[:STATe] {ON | OFF | 1 | 0}	DISPlay[:WINDow<Wnd>]:TITLe[:STATe]
DISPlay:WINDow<Ch>:TRACe<Tr>:MEMory[:STATe] {ON | OFF | 1 | 0}	TRACe:COPY
DISPlay:WINDow{1...4}:TRACe{1...7}:Y[:SCALe]:AUTO	DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:Y[:SCALe]:AUTO
DISPlay:WINDow{1...4}:Y[:SCALe]:DIVisions <numeric>	DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:Y[:SCALe]:PDIVision
DISPlay:WINDow{1...4}:Y[:SCALe]:RLEVel <numeric>	DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:Y[:SCALe]:RLEVel
DISPlay:WINDow{1...4}:TRACe{1...7}:Y[:SCALe]:RPOSition <numeric>	DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:Y[:SCALe]:RPOSition
FORMat:BORDer {NORMal | SWAPped}	FORMat:BORDer
FORMat:DATA {ASCii | REAL}	FORMat[:DATA]
INITiate{1...4}:CONTinuous {ON | OFF | 1 | 0}	INITiate<Ch>:CONTinuous
INITiate{1...4}[:IMMediate]	INITiate<Ch>[:IMMediate][:DUMMy]
MMEMory:CATalog? <string>	MMEMory:CATalog?
MMEMory:COPY <string 1> <string 2>	MMEMory:COPY
MMEMory:DELete <string>	MMEMory:DELete
MMEMory:LOAD:LIMit <string>	MMEMory:LOAD:LIMit
MMEMory:LOAD:SEGMent <string>	MMEMory:LOAD:SEGMent
MMEMory:LOAD:[:STATe] <string>	MMEMory:LOAD:STATe
MMEMory:MDIRectory <string>	MMEMory:MDIRectory
MMEMory:STORe[:STATe] <string>	MMEMory:STORe:STATe
MMEMory:STORe:FDATa <string>	MMEMory:STORe:TRACe

ENA Emulation Command	R&S ZNB Default Parser Command
MMEMory:STORe:LIMit <string>	MMEMory:STORe:LIMit
MMEMory:STORe:SEGMent <string>	MMEMory:STORe:SEGMent
MMEMory:STORe:SNP:DATA <filename>	MMEMory:STORe:TRACe:PORTs
MMEMory:STORe:SNP:TYPE:S1P <numeric>	MMEMory:STORe:TRACe:PORTs
MMEMory:STORe:SNP:TYPE:S2P <numeric1>, <numeric2>	MMEMory:STORe:TRACe:PORTs
MMEMory:STORe:SNP:TYPE:S3P <numeric1>, <numeric2>, <numeric3>	MMEMory:STORe:TRACe:PORTs
MMEMory:STORe:SNP:TYPE:S4P <numeric3>, <numeric2>, <numeric3>, <numeric4>	MMEMory:STORe:TRACe:PORTs
MMEMory:STORe:STYPe {STATe | CSTate | DSTate | CDSTate}	MMEMory:STORe:STATe Always saves measurement conditions and calibration state (CSTate), regardless of what you set it to.
SENSe{1...4}:AVERage:CLEar	[SENSe<Ch>:]AVERage:CLEar
SENSe{1...4}:AVERage:COUNt <numeric>	[SENSe<Ch>:]AVERage:COUNt
SENSe{1...4}:AVERage[:STATe] {ON | OFF | 1 | 0}	[SENSe<Ch>:]AVERage[:STATe]
SENSe{1...4}:BWIDth[:RESolution] <bandwidth>	[SENSe<Ch>:]BWIDth[:RESolution]
SENSe{1...4}:BWIDth[:RESolution]: SELect NORMal | HIGH	[SENSe<Ch>:]BWIDth[:RESolution]:SELect
SENSe{1...4}:CORRection:COLLect[: ACQuire]:LOAD <numeric> SENSe{1...4}:CORRection:COLLect[: ACQuire]:OPEN <numeric> SENSe{1...4}:CORRection:COLLect[: ACQuire]:SHORt <numeric> SENSe{1...4}:CORRection:COLLect[: ACQuire]:ISOLation <numeric1>, <numeric2> SENSe{1...4}:CORRection:COLLect[: ACQuire]:THRU <numeric1>, <numeric2>	[SENSe<Ch>:]CORRection:COLLect[: ACQuire]:SELected
SENSe{1..4}:CORRection:COLLect:CKIT[: SELect] <integer>	[SENSe:]CORRection:CKIT:SELect
SENSe{1...4}:CORRection:COLLect:ECAL: ISOLation[:STATe] {OFF | 0}	None (no action)
SENSe{1..4}:CORRection:COLLect:ECAL: PATH <EcalPort>, <VnaPort>	[SENSe<Ch>:]CORRection:COLLect:AUTO: PORTs

ENA Emulation Command	R&S ZNB Default Parser Command
SENSe{1...4}:CORRection:COLLect:ECAL:SOLT1 <numeric> SENSe{1...4}:CORRection:COLLect:ECAL:SOLT2 <numeric1>, <numeric2> SENSe{1...4}:CORRection:COLLect:ECAL:SOLT3 <numeric1>, <numeric2>, <numeric3> SENSe{1...4}:CORRection:COLLect:ECAL:SOLT4 1,2,3,4	[SENSe<Ch>:]CORRection:COLLect:AUTO:TYPE
SENSe{1...4}:CORRection:COLLect:ECAL:UTHR[:STATe] {OFF | 0}	
SENSe<ch>:CORRection:COLLect:GUIDed:CKIT:PORT<pt>:CATalog?	[SENSe:]CORRection:CKIT:CATalog?
SENSe{1...4}:CORRection:COLLect:METHod[:RESPonse]:OPEN <numeric> SENSe{1...4}:CORRection:COLLect:METHod[:RESPonse]:SHORt <numeric> SENSe{1...4}:CORRection:COLLect:METHod:SOLT1 <numeric> SENSe{1...4}:CORRection:COLLect:METHod:SOLT2 <numeric1>, <numeric2> SENSe{1...4}:CORRection:COLLect:METHod:SOLT3 <numeric1>, <numeric2>, <numeric3> SENSe{1...4}:CORRection:COLLect:METHod:SOLT4 1,2,3,4 SENSe{1...4}:CORRection:COLLect:METHod[:RESPonse]:THRU <numeric1>, <numeric2> SENSe{1...4}:CORRection:COLLect:METHod:TYPE? SENSe{1...4}:CORRection:TYPE{1...4}?	[SENSe<Ch>:]CORRection:COLLect:METHod:DEFine
SENSe{1...4}:CORRection:COLLect:SAVE	[SENSe<Ch>:]CORRection:COLLect:SAVE[:DUMMy]
SENSe{1..4}:CORRection:COLLect:UTHRu[:STATe] {OFF | 0}	
SENSe<Ch>:CORRection:EXTension[:STATe] {ON | OFF | 1 | 0}	[SENSe<Ch>:]CORRection:OFFSet<PhyPt>:COMPensation[:STATe]
SENSe{1...4}:CORRection:EXTension:PORT{[1]|2 | 3|4}:TIME <numeric>	[SENSe<Ch>:]CORRection:EDELay<PhyPt>[:TIME]
SENSe{1...4}:CORRection:STATe {ON | OFF | 1 | 0}	[SENSe<Ch>:]CORRection[:STATe]
SENSe{1...4}:FREQuency:CENTer <numeric>	[SENSe<Ch>:]FREQuency:CENTer
SENSe{1...4}:FREQuency:DATA?	CALCulate<Chn>:DATA:STIMulus?
SENSe{1...4}:FREQuency:SPAN <numeric>	[SENSe<Ch>:]FREQuency:SPAN

ENA Emulation Command	R&S ZNB Default Parser Command
SENSe{1...4}:FREQuency:STARt <numeric>	[SENSe<Ch>:]FREQuency:STARt
SENSe{1...4}:FREQuency:STOP <numeric>	[SENSe<Ch>:]FREQuency:STOP
SENSe{1...4}:ROSCillator:SOURce?	[SENSe:]ROSCillator[:SOURce]
SENSe{1...4}:SEGMent:SWEep:POINts?	[SENSe<Ch>:]SEGMent<Seg>:SWEep:POINts
SENSe{1...4}:SEGMent:SWEep:TIME?	[SENSe<Ch>:]SEGMent<Seg>:SWEep:TIME
SENSe{1...4}:SEGMent:DATA 5,<mode>, <ifbw>, <pow>, <del>, <time>, <segm>, <star 1>, <stop 1>, <nop 1>, <ifbw 1>, <pow 1>, <del 1>, <time 1>,...,	
SENSe{1...4}:SWEep:DELay <numeric>	[SENSe<Ch>:]SWEep:DWELl
SENSe{1...4}:SWEep:MOD CONTinuous | HOLD | SINGle SENSe{1...4}:SWEep:MODE CONTinuous | HOLD | SINGle	
SENSe{1...4}:SWEep:POINts <numeric>	[SENSe<Ch>:]SWEep:POINts
SENSe{1...4}:SWEep:TIME[:DATA] <numeric>	[SENSe<Ch>:]SWEep:TIME
SENSe{1...4}:SWEep:TIME:AUTO {ON | OFF | 1 | 0}	[SENSe<Ch>:]SWEep:TIME:AUTO
SENSe{1...4}:SWEep:TYPE {LINear | SEGMent}	[SENSe<Ch>:]SWEep:TYPE
SERVice:CHANnel:COUNt?	n.a. (no limit)
SERVice:CHANnel:TRACe:COUNt?	n.a. (no limit)
SOURce{1...4}:POWer[:LEVel][:IMMediate][:AMPLitude] <numeric>	SOURce<Ch>:POWer<PhyPt>[:LEVel][:IMMediate][:AMPLitude]
SOURce<ch>:POWer<pt>:CORRection[:STATe] {ON | OFF | 1 | 0}	SOURce<Ch>:POWer<PhyPt>:CORRection:STATe
SOURce<ch>:POWer<pt>:CORRection:COLLect:AVERage[:COUNt] <numeric>	SOURce:POWer:CORRection:COLLect:AVERage[:COUNt]
SOURce:POWer<pt>:CORRection:COLLect:AVERage:NTOLerance <numeric>	SOURce:POWer:CORRection:COLLect:AVERage:NTOLerance
SOURce<ch>:POWer<pt>:CORRection:COLLect:SAVE [<RREC>]	n.a. (instrument automatically applies calibration after it is complete)
SOURce{1..4}:POWer:PORT:COUPle {ON | 1}	
STATus:OPERation:CONDition?	STATus:QUEStionable:CONDition?
STATus:OPERation:ENABle <numeric>	STATus:QUEStionable:ENABle
STATus:OPERation[:EVENt]?	STATus:QUEStionable[:EVENt]?
STATus:QUEStionable:NTRansition <numeric>	STATus:QUEStionable:NTRansition

ENA Emulation Command	R&S ZNB Default Parser Command
STATus:QUEStionable:PTRansition <numeric>	STATus:QUEStionable:LIMit<Lev>:PTRansition
STATus:QUEStionable:LIMit:CONDition?	STATus:QUEStionable:LIMit<Lev>:CONDition?
STATus:QUEStionable:LIMit:ENABle <numeric>	STATus:QUEStionable:LIMit<Lev>:ENABle
STATus:QUEStionable:LIMit[:EVENt]?	STATus:QUEStionable:LIMit<Lev>[:EVENt]?
STATus:QUEStionable:LIMit:NTRansition <numeric>	STATus:QUEStionable:LIMit<Lev>:NTRansition
STATus:QUEStionable:LIMit:PTRansition <numeric>	STATus:QUEStionable:LIMit<Lev>:PTRansition
SYSTem:KLOCk:KBD {ON | OFF | 1 | 0}	SYSTem:KLOCk
TRIGger[:SEQuence]:SOURce {INTernal | EXTernal | MANual | BUS}	TRIGger<Ch>[:SEQuence]:SOURce

Glossary: Frequently used terms

A

Active channel: Channel belonging to the active trace. The active channel is highlighted in the channel list below the diagram. The active channel is not relevant in remote control where each channel can contain an active trace.

Active marker: Marker that can be changed using the settings of the Marker menu (Delta Mode, Ref. Mkr -> Mkr, Mkr Format). The active marker is also used for the Marker Functions. It appears in the diagram with an enlarged marker symbol and font size and with a dot placed in front of the marker line in the info field.

Active menu: The menu containing the last executed command. If the softkey bar is displayed (Display - Config./View - Softkey Labels on), then the active menu is indicated on top of the softkey bar.

Active trace (manual control): Trace that is selected to apply the settings in the Trace menu. The active trace is highlighted in the trace list of the active diagram area. It can be different from the active trace in remote control.

Active trace (remote control): One trace of each channel that has been selected as the active trace (`CALCulate<Ch>:PARameter:SELect <trace name>`). Many commands (e.g. `TRACE...`) act on the active trace. It can be different from the active trace in manual control.

C

Cal pool: The cal pool is a collection of correction data sets (cal groups) that the analyzer stores in a common directory. Cal groups in the pool can be applied to different channels and recall sets.

Calibration: The process of removing systematic errors from the measurement (system error correction). See also TOSM, TOM, TRM, TRL, TNA...

Calibration kit: Set of physical calibration standards for a particular connector family.

Calibration standard: Physical device that has a known or predictable magnitude and phase response within a given frequency range. Calibration standards are grouped into several types (open, through, match,...) corresponding to the different input quantities for the analyzer's error models.

Calibration unit: Integrated solution for automatic calibration of multiple ports (accessories R&S ZV-Zxx, R&S ZN-Z5x and R&S ZN-Z15x). The unit contains calibration standards that are electronically switched when a calibration is performed.

Channel: A channel contains hardware-related settings to specify how the network analyzer collects data. Each channel is stored in an independent data set. The channel

settings complement the definitions of the Trace menu; they apply to all traces assigned to the channel.

Compression point: The x-dB compression point of an S-parameter or ratio is the stimulus signal level where the magnitude of the measured quantity has dropped by x dB compared to its value at small stimulus signal levels (small-signal value).

Confirmation dialog box: Standard dialog box that pops up to display an error message or a warning. The current action can be either continued (OK) or canceled (Cancel) on closing the dialog box.

Crosstalk: The occurrence of a signal at the receive port of the analyzer which did not travel through the test setup and the DUT but leaks through other internal paths. Crosstalk causes an isolation error in the measurement which can be corrected by means of a calibration.

CW frequency: Continuous Wave frequency; fixed stimulus frequency used in Power, CW Time and CW Mode sweeps.

D

Data trace: Trace filled with measurement data and updated after each sweep (dynamic trace).

Diagram area: Rectangular portion of the screen used to display traces. Diagram areas are arranged in windows; they are independent of trace and channel settings.

Directivity error: Measurement error caused by a coupler or bridge in the analyzer's source port causing part of the generated signal to leak through the forward path into the receive path instead of being transmitted towards the DUT. The directivity error can be corrected by means of a full one port calibration or one of the two-port calibration methods (except normalization).

Discrete marker: The stimulus value of a discrete marker always coincides with a sweep point so that the marker does not show interpolated measurement values.

DUT: Device under test; generic term for any electrical device or circuit which the vector network analyzer can measure. Typical DUTs are filters, amplifiers, or mixers.

E

Excursion: Difference between the response values at a local maximum (minimum) of the trace and at the two closest local minima (maxima) to the left and to the right.

Extrapolation: Calculation of a numeric value for a new sweep point outside the original sweep range from the numeric values of the existing sweep points. The analyzer can extrapolate calibration data, transmission coefficients etc. to extend the sweep range. If not otherwise stated, the numeric value of the first (last) sweep point is

assigned to all new points below (above) the original sweep range. See also --> interpolation.

F

Forward: A measurement on a two-port DUT is said to be in forward direction if the source signal (stimulus) is applied to port 1 of the DUT.

H

Harmonic: Integer multiple of the fundamental frequency. The fundamental is the first harmonic, the nth harmonic is n times the frequency of the fundamental.

Harmonic distortion: The production of harmonic frequencies (harmonics) by an electronic system when a signal is applied at the input.

Harmonic grid: A set of equidistant frequency points f_i (I = 1...n) with spacing Delta(f) and the additional condition that f_1 = Delta(f). A harmonic grid is required for low pass time domain transforms.

I

Intercept point: Fictitious lower-tone DUT input/output level where the intermodulation suppression (-->) for a given intermodulation product reaches 0 dB.

Intermodulation measurement: Measurement where the DUT is supplied with two RF signals of equal power but different frequencies termed the upper and lower tone. The analyzer measures the frequency-converting behavior of the DUT (--> intermodulation product).

Intermodulation product: Special type of emissions of a nonlinear DUT that is supplied with a two-tone RF signal (--> intermodulation measurement). The intermodulation products occur at frequencies which correspond to sums and differences of the upper and lower tone frequencies and their integer multiples.

Intermodulation suppression: The ratio of the power of an --> intermodulation product to the power of the lower tone fundamental wave.

Interpolation: Calculation of a numeric value for a specific sweep point from the numeric values of the adjacent points. The analyzer can interpolate calibration data, transmission coefficients etc. to account for a modified set of sweep points. If not otherwise stated, linear interpolation is used. See also --> extrapolation.

Isolation error: Measurement error caused by a crosstalk between the source and receive port of the analyzer.

L

Limit check: Comparison of measurement results with user-defined limits, and display of a pass/fail indication.

A TTL signal can be generated in addition, if a limit is violated.

Limit line: A limit line is a set of data to specify the allowed range for some or all points of a trace. Typically, limit lines are used to check whether a DUT conforms to the rated specifications (conformance testing).

Load match error: Measurement error caused by a mismatch of the analyzer's receive (load) port causing part of the signal transmitted through the DUT to be reflected off the receive port so that it is not measured there. The load match error can be corrected by means of a two-port calibration (except normalization).

M

Marker: Tool for selecting points on the trace and for numerical readout of measured data. A marker is displayed with a symbol (a triangle, a crossbar or a line) on the trace; its coordinates are shown in the marker info field.

Mathematical trace: Trace that is calculated according to a mathematical expression, e.g. the one defined in the Define Math dialog. The expression is a mathematical relation between constants and the data or memory traces of the active recall set.

Measurement point: Result of the measurement at a specified stimulus value (frequency/power/time).

Measurement result: Set of all measurement points acquired in a measurement (e.g. a sweep). The measurement result is displayed in a diagram area and forms a trace.

Memory trace: Trace that is associated to a data trace and stored in the memory. Data traces and the associated memory traces share the same channel and scale settings. Alternatively, memory traces can be imported from a file.

Mixer: Device that converts an RF signal at one frequency into a signal at another frequency. The frequency that is to be shifted is applied at the RF input and the frequency shifting signal (from a local oscillator, LO) is applied to the RF mixer's LO port, resulting in an output signal at the mixer's Intermediate Frequency (IF) port.

P

Partial measurement: Measurement at a specified stimulus value maintaining definite hardware settings. Depending on the measurement type, several partial measurements may be needed to obtain a measurement point. A full n-port S-parameter measurement requires n partial measurements with n different drive ports.

Peak: Local maximum or local minimum (dip) on the trace. In the Trace - Search menu, it is possible to define a minimum excursion that both types of peaks must have to be considered valid.

R

Recall Set: A recall set comprises a set of diagram areas with all displayed information that can be stored to a VNA recall set file (`*.znx|*.znxml`). Each recall set is displayed in an independent tab.

Reflection tracking error: Frequency-dependent variation of the ratio of the reflected wave to the reference wave at a test port when an ideal reflection coefficient (= 1) is measured. The reflection tracking error can be corrected by means of a reflection normalization or one of the more sophisticated calibration methods.

Reverse: A measurement on a two-port DUT is said to be in reverse direction if the source signal (stimulus) is applied to port 2 of the DUT.

S

Source match error: Measurement error caused by a mismatch of the analyzer's source port causing part of the signal reflected off the DUT to be reflected again off the source port so that it is not measured there. The source match error can be corrected by means of a full one-port calibration or a two-port calibration (except normalization).

Stimulus value: Value of the sweep variable (frequency/power/time/point number) where a measurement is taken. Also termed sweep point.

Sweep: Series of consecutive measurements taken at a specified sequence of stimulus values = series of consecutive measurement points.

Sweep point: Value of the sweep variable (stimulus value: frequency/power/time) where a measurement is taken.

Sweep range: Continuous range of the sweep variable (frequency/power/time) containing the sweep points where the analyzer takes measurements. In a Segmented Frequency sweep the sweep range can be composed of several parameter ranges or single points.

Sweep segment: Continuous frequency range or single frequency point where the analyzer measures at specified instrument settings (generator power, IF bandwidth etc.). In the Segmented Frequency sweep type the entire sweep range can be composed of several sweep segments.

T

TNA: A calibration type using a Through, a symmetric Network and an Attenuation standard. The properties of the Network and the Attenuation don't have to be known exactly. Like TRL and TRM, TNA is especially useful for DUTs in planar line technology.

TOM: A calibration type using three fully known standards (Through, Open, Match), recommended for 2-port measurements on coaxial systems.

Topology: Assignment of the physical ports of the VNA to the logical ports used for the measurement of mixed mode S-parameters (balance-unbalance conversion).

TOSM: A calibration type using a Through plus the one-port standards Open, Short, Match, to be connected to each calibrated port. Classical 12-term error model, also referred to as SOLT. See also UOSM.

TRL: A calibration type using the two-port standards Through and Line, which are both assumed to be ideally matched. Beyond that, the Through must be lossless, and its length must be exactly known. Especially useful for DUTs in planar line technology.

TRM: A calibration type which requires a low-reflection, low-loss Through standard with an electrical length that may be different from zero, a Reflect, and a Match. Especially useful for DUTs in test fixtures.

TSM: A calibration type using three fully known standards (Through, Short, Match), recommended for 2-port measurements on coaxial systems.

U

UOSM: A variant of TOSM calibration using an unknown but reciprocal Through standard. Especially for port combinations with different connector types.

V

VNA: (Vector) Network Analyzer, in particular the R&S ZNB.

List of commands

[SENSe:]CDLL:ADD....1311
[SENSe:]CDLL:LIST:TASK?....1312
[SENSe:]CDLL:LIST?....1312
[SENSe:]CDLL:REMove....1313
[SENSe:]CORRection:CKIT:<ConnType>:LSELect....1327
[SENSe:]CORRection:CKIT:<ConnType>:SELect....1322
[SENSe:]CORRection:CKIT:<OnePortStandardType>:WLABel:SDATa?....1328
[SENSe:]CORRection:CKIT:<StandardType>....1315
[SENSe:]CORRection:CKIT:<StandardType>:WLABel....1328
[SENSe:]CORRection:CKIT:<TwoPortStandard>:WLABel:SDATa?....1329
[SENSe:]CORRection:CKIT:ADD....1319
[SENSe:]CORRection:CKIT:CATalog?....1319
[SENSe:]CORRection:CKIT:COPY....1320
[SENSe:]CORRection:CKIT:DELete....1321
[SENSe:]CORRection:CKIT:DMODe....1321
[SENSe:]CORRection:CKIT:INSTall....1633
[SENSe:]CORRection:CKIT:LABel....1321
[SENSe:]CORRection:CKIT:LCATalog?....1325
[SENSe:]CORRection:CKIT:LDELete....1325
[SENSe:]CORRection:CKIT:LLABel....1325
[SENSe:]CORRection:CKIT:LSELect....1326
[SENSe:]CORRection:CKIT:SELect....1322
[SENSe:]CORRection:CKIT:STANdard:CATalog?....1323
[SENSe:]CORRection:CKIT:STANdard:DATA?....1323
[SENSe:]CORRection:CKIT:STANdard:LCATalog?....1328
[SENSe:]CORRection:CKIT:THRough:SELect....1324
[SENSe:]CORRection:COLLect:AUTO:CKIT....1342
[SENSe:]CORRection:COLLect:AUTO:CKIT:PASSword....1343
[SENSe:]CORRection:COLLect:AUTO:CKIT:PORTs....1343
[SENSe:]CORRection:COLLect:AUTO:CKIT:PORTs:ADD....1344
[SENSe:]CORRection:COLLect:AUTO:EXPLicit....1347
[SENSe:]CORRection:COLLect:AUTO:MCONnect....1348
[SENSe:]CORRection:COLLect:AUTO:PORTs:CONNection?....1349
[SENSe:]CORRection:COLLect:AVERage....1355
[SENSe:]CORRection:COLLect:CHANnels:ALL....1355
[SENSe:]CORRection:COLLect:CHANnels:MCTYpes....1355
[SENSe:]CORRection:COLLect:FIXTure:LMParameter:LOSS[:STATe]....1365
[SENSe:]CORRection:COLLect:FIXTure:LMParameter[:STATe]....1364
[SENSe:]CORRection:COLLect:PMETer:ID....1369
[SENSe:]CORRection:COLLect:SWEep:CONTinuous....1371
[SENSe:]CORRection:CONNection....1378
[SENSe:]CORRection:CONNection:CATalog?....1380
[SENSe:]CORRection:EDELay:VNETwork....1386
[SENSe:]CORRection:METas:CABLe:CATalog?....1372
[SENSe:]ROSCillator:EXTernal:FREQuency....1431
[SENSe:]ROSCillator[:SOURce]....1432
[SENSe:]SNPMeasure:DISPlay:CMAP<pos>:RGB....1455

[SENSe:]SNPMeasure:DISPlay:RST....1456
[SENSe:]SNPMeasure:DUT:IMPort....1456
[SENSe:]SNPMeasure:DUT:INIT:IDEalize....1457
[SENSe:]SNPMeasure:DUT:INIT:MEASure....1457
[SENSe:]SNPMeasure:DUT:INIT:RANGe....1457
[SENSe:]SNPMeasure:DUT:NUMBer....1458
[SENSe:]SNPMeasure:DUT:PART:ADD....1459
[SENSe:]SNPMeasure:DUT:PART<pos>:DELete....1460
[SENSe:]SNPMeasure:DUT:PART<pos>:MOVE....1460
[SENSe:]SNPMeasure:DUT:PART<pos>:PORT....1460
[SENSe:]SNPMeasure:DUT:PART<pos>:WIRE....1461
[SENSe:]SNPMeasure:DUT:PART<pos>:WIRE:IDENtity....1462
[SENSe:]SNPMeasure:DUT:PART<pos>:WIRE:POLarity....1462
[SENSe:]SNPMeasure:DUT:PART<pos>?....1459
[SENSe:]SNPMeasure:DUT:PREFix....1463
[SENSe:]SNPMeasure:DUT:RIMPedance....1463
[SENSe:]SNPMeasure:DUT:TEMPlate....1463
[SENSe:]SNPMeasure:DUT?....1456
[SENSe:]SNPMeasure:EXPort....1463
[SENSe:]SNPMeasure:INSTrument:PORT....1464
[SENSe:]SNPMeasure:MINTegrity....1464
[SENSe:]SNPMeasure:MINTegrity:RESTore....1465
[SENSe:]SNPMeasure:RECall....1465
[SENSe:]SNPMeasure:RST....1465
[SENSe:]SNPMeasure:SAVE....1465
[SENSe:]SNPMeasure:STATus?....1466
[SENSe:]SNPMeasure:STEP<pos>:CLEar....1467
[SENSe:]SNPMeasure:STEP<pos>:IMPort....1468
[SENSe:]SNPMeasure:STEP<pos>:MEASure....1468
[SENSe:]SNPMeasure:STEP<pos>?....1466
[SENSe:]SWEep:COUNt:ALL....1471
[SENSe:]UDSParams:ACTive....1482
[SENSe:]UDSParams<Pt>:PARam....1482
[SENSe<Ch>:]AVERage:CLEar....1309
[SENSe<Ch>:]AVERage:COUNt....1309
[SENSe<Ch>:]AVERage:MODE....1309
[SENSe<Ch>:]AVERage[:STATe]....1308
[SENSe<Ch>:]BANDwidth[:RESolution]....1310
[SENSe<Ch>:]BANDwidth[:RESolution]:SELect....1311
[SENSe<Ch>:]BWIDth[:RESolution]....1310
[SENSe<Ch>:]BWIDth[:RESolution]:SELect....1311
[SENSe<Ch>:]CDLL:PERManent:ADDitional....1312
[SENSe<Ch>:]CDLL:PERManent:TASK....1313
[SENSe<Ch>:]CDLL:PERManent[:STATe]....1313
[SENSe<Ch>:]CDLL:TASK:ADDRess....1314
[SENSe<Ch>:]CDLL[:STATe]....1314
[SENSe<Ch>:]CORRection:CDATa....1376
[SENSe<Ch>:]CORRection:CDATa:PORT<PhyPt>....1376
[SENSe<Ch>:]CORRection:CKIT:<ConnType>:<StandardType>....1632
[SENSe<Ch>:]CORRection:COLLect:AUTO....1333

[SENSe<Ch>:]CORRection:COLLect:AUTO:ASSignment:ALL:COUNt? 1335
[SENSe<Ch>:]CORRection:COLLect:AUTO:ASSignment:COUNt? 1336
[SENSe<Ch>:]CORRection:COLLect:AUTO:ASSignment:DEFine:DEFault 1339
[SENSe<Ch>:]CORRection:COLLect:AUTO:ASSignment:DEFine:OPTimized 1340
[SENSe<Ch>:]CORRection:COLLect:AUTO:ASSignment:DEFine:TPORt:DEFault 1341
[SENSe<Ch>:]CORRection:COLLect:AUTO:ASSignment:DELete:ALL 1342
[SENSe<Ch>:]CORRection:COLLect:AUTO:ASSignment<Asg>:ACQuire 1334
[SENSe<Ch>:]CORRection:COLLect:AUTO:ASSignment<Asg>:CHECk? 1334
[SENSe<Ch>:]CORRection:COLLect:AUTO:ASSignment<Asg>:DEFine 1336
[SENSe<Ch>:]CORRection:COLLect:AUTO:ASSignment<Asg>:DEFine:TPORt 1341
[SENSe<Ch>:]CORRection:COLLect:AUTO:CONFigure 1345
[SENSe<Ch>:]CORRection:COLLect:AUTO:CPORt 1347
[SENSe<Ch>:]CORRection:COLLect:AUTO:PORTs 1348
[SENSe<Ch>:]CORRection:COLLect:AUTO:PORTs:TYPE 1350
[SENSe<Ch>:]CORRection:COLLect:AUTO:POWer 1351
[SENSe<Ch>:]CORRection:COLLect:AUTO:POWer:DEFault 1351
[SENSe<Ch>:]CORRection:COLLect:AUTO:REPeat 1352
[SENSe<Ch>:]CORRection:COLLect:AUTO:SAVE 1352
[SENSe<Ch>:]CORRection:COLLect:AUTO:TEMPcomp 1353
[SENSe<Ch>:]CORRection:COLLect:AUTO:TERMination 1353
[SENSe<Ch>:]CORRection:COLLect:AUTO:TYPE 1353
[SENSe<Ch>:]CORRection:COLLect:AUTO:UTHRough 1354
[SENSe<Ch>:]CORRection:COLLect:CKIT:INSTall 1357
[SENSe<Ch>:]CORRection:COLLect:CKIT:LOAD 1357
[SENSe<Ch>:]CORRection:COLLect:CKIT:PORT<PhyPt>? 1357
[SENSe<Ch>:]CORRection:COLLect:CONNection:GENDers 1359
[SENSe<Ch>:]CORRection:COLLect:CONNection:PORTs 1360
[SENSe<Ch>:]CORRection:COLLect:CONNection<PhyPt> 1358
[SENSe<Ch>:]CORRection:COLLect:DELete 1360
[SENSe<Ch>:]CORRection:COLLect:DETector 1361
[SENSe<Ch>:]CORRection:COLLect:DISCard 1361
[SENSe<Ch>:]CORRection:COLLect:FIXTure:EXPort 1363
[SENSe<Ch>:]CORRection:COLLect:FIXTure:IMPort 1363
[SENSe<Ch>:]CORRection:COLLect:FIXTure:SAVE 1365
[SENSe<Ch>:]CORRection:COLLect:FIXTure:STARt 1365
[SENSe<Ch>:]CORRection:COLLect:FIXTure[:ACQuire] 1362
[SENSe<Ch>:]CORRection:COLLect:LOAD:SELected 1366
[SENSe<Ch>:]CORRection:COLLect:METHod 1634
[SENSe<Ch>:]CORRection:COLLect:METHod:DEFine 1367
[SENSe<Ch>:]CORRection:COLLect:SAVE:DEFault 1635
[SENSe<Ch>:]CORRection:COLLect:SAVE:SELected:DEFault 1369
[SENSe<Ch>:]CORRection:COLLect:SAVE:SELected[:DUMMy] 1370
[SENSe<Ch>:]CORRection:COLLect:SAVE[:DUMMy] 1636
[SENSe<Ch>:]CORRection:COLLect:SCONnection<PhyPt> 1371
[SENSe<Ch>:]CORRection:COLLect[:ACQuire] 1633
[SENSe<Ch>:]CORRection:COLLect[:ACQuire]:SELected 1331
[SENSe<Ch>:]CORRection:CONNection:DELete 1380
[SENSe<Ch>:]CORRection:DATA 1637
[SENSe<Ch>:]CORRection:DATA:PARameter:COUNt? 1383
[SENSe<Ch>:]CORRection:DATA:PARameter<Sfk>:PORT<PhyPt>? 1380

[SENSe<Ch>:]CORRection:DATA:PARameter<Sfk>? 1380
[SENSe<Ch>:]CORRection:DATE? 1383
[SENSe<Ch>:]CORRection:DELete 1384
[SENSe<Ch>:]CORRection:EDELay:AUTO 1384
[SENSe<Ch>:]CORRection:EDELay<PhyPt>:DIELectric 1384
[SENSe<Ch>:]CORRection:EDELay<PhyPt>:DISTance 1384
[SENSe<Ch>:]CORRection:EDELay<PhyPt>:ELENgth 1385
[SENSe<Ch>:]CORRection:EDELay<PhyPt>:VELocity 1386
[SENSe<Ch>:]CORRection:EDELay<PhyPt>[:TIME] 1385
[SENSe<Ch>:]CORRection:LOSS:AUTO 1387
[SENSe<Ch>:]CORRection:LOSS<PhyPt> 1386
[SENSe<Ch>:]CORRection:LOSS<PhyPt>:FREQuency 1387
[SENSe<Ch>:]CORRection:LOSS<PhyPt>:OFFSet 1388
[SENSe<Ch>:]CORRection:LOSS<PhyPt>:SECond 1386
[SENSe<Ch>:]CORRection:LOSS<PhyPt>:SECond:FREQuency 1387
[SENSe<Ch>:]CORRection:LOSS<PhyPt>:SECond:STATe 1388
[SENSe<Ch>:]CORRection:METas:CABLe:MOVement 1372
[SENSe<Ch>:]CORRection:METas:CABLe:REConnection 1373
[SENSe<Ch>:]CORRection:METas:CABLe:TYPE 1373
[SENSe<Ch>:]CORRection:METas:UNCertainty 1374
[SENSe<Ch>:]CORRection:METas[:STATe] 1374
[SENSe<Ch>:]CORRection:OFFSet<PhyPt>:COMPensation[:STATe] 1389
[SENSe<Ch>:]CORRection:OFFSet<PhyPt>:DFComp[:STATe]? 1389
[SENSe<Ch>:]CORRection:OFFSet<PhyPt>:MAGNitude 1639
[SENSe<Ch>:]CORRection:OFFSet<PhyPt>[:STATe] 1389
[SENSe<Ch>:]CORRection:PCAL 1390
[SENSe<Ch>:]CORRection:POWer:DATA 1392
[SENSe<Ch>:]CORRection:POWer:DATA:PORT<PhyPt> 1392
[SENSe<Ch>:]CORRection:POWer<PhyPt>:ACQuire 1391
[SENSe<Ch>:]CORRection:POWer<PhyPt>:AWAVe[:STATe] 1392
[SENSe<Ch>:]CORRection:POWer<PhyPt>:IMODulation:ACQuire 1393
[SENSe<Ch>:]CORRection:POWer<PhyPt>[:STATe] 1394
[SENSe<Ch>:]CORRection:SMATrix:CDATa 1395
[SENSe<Ch>:]CORRection:SMATrix:CDATa:PORT<PhyPt> 1395
[SENSe<Ch>:]CORRection:STIMulus:PORT<PhyPt>? 1396
[SENSe<Ch>:]CORRection:STIMulus? 1396
[SENSe<Ch>:]CORRection[:STATe] 1396
[SENSe<Ch>:]COUPle 1480
[SENSe<Ch>:]DC<DCInp>:RANGe 1480
[SENSe<Ch>:]FREQuency:CENTer 1398
[SENSe<Ch>:]FREQuency:CONVersion 1399
[SENSe<Ch>:]FREQuency:CONVersion:ARBitrary 1399
[SENSe<Ch>:]FREQuency:CONVersion:ARBitrary:PMETer<Pmtr> 1400
[SENSe<Ch>:]FREQuency:CONVersion:GAIN:LMCorrection 1401
[SENSe<Ch>:]FREQuency:CONVersion:MIXer:FIXed<Stg> 1401
[SENSe<Ch>:]FREQuency:CONVersion:MIXer:FUNDamental 1402
[SENSe<Ch>:]FREQuency:CONVersion:MIXer:IFFixed 1639
[SENSe<Ch>:]FREQuency:CONVersion:MIXer:IFPort 1402
[SENSe<Ch>:]FREQuency:CONVersion:MIXer:LOFixed 1640
[SENSe<Ch>:]FREQuency:CONVersion:MIXer:LOMultiplier<Stg> 1403

[SENSe<Ch>:]FREQuency:CONVersion:MIXer:LOPort<Stg> 1403
[SENSe<Ch>:]FREQuency:CONVersion:MIXer:MFFixed 1404
[SENSe<Ch>:]FREQuency:CONVersion:MIXer:RFFixed 1640
[SENSe<Ch>:]FREQuency:CONVersion:MIXer:RFMultiplier 1405
[SENSe<Ch>:]FREQuency:CONVersion:MIXer:RFPort 1406
[SENSe<Ch>:]FREQuency:CONVersion:MIXer:STAGes 1406
[SENSe<Ch>:]FREQuency:CONVersion:MIXer:TFRequency<Stg> 1406
[SENSe<Ch>:]FREQuency:FIXed 1407
[SENSe<Ch>:]FREQuency:IMODulation:CONVersion 1408
[SENSe<Ch>:]FREQuency:IMODulation:CWFixed 1408
[SENSe<Ch>:]FREQuency:IMODulation:LTONe 1408
[SENSe<Ch>:]FREQuency:IMODulation:ORDer<Im>[:STATe] 1409
[SENSe<Ch>:]FREQuency:IMODulation:RECeiver 1409
[SENSe<Ch>:]FREQuency:IMODulation:SPECtrum:MORDer 1410
[SENSe<Ch>:]FREQuency:IMODulation:SPECtrum[:STATe] 1410
[SENSe<Ch>:]FREQuency:IMODulation:TDIStance 1411
[SENSe<Ch>:]FREQuency:IMODulation:UTONe 1412
[SENSe<Ch>:]FREQuency:MODE 1641
[SENSe<Ch>:]FREQuency:SBANd 1412
[SENSe<Ch>:]FREQuency:SEGMent:AXIS 1413
[SENSe<Ch>:]FREQuency:SPAN 1414
[SENSe<Ch>:]FREQuency:STARt 1414
[SENSe<Ch>:]FREQuency:STOP 1414
[SENSe<Ch>:]FREQuency[:CW] 1407
[SENSe<Ch>:]HARMonic:AUTO 1415
[SENSe<Ch>:]HARMonic:DLENgth:DATA 1415
[SENSe<Ch>:]HARMonic:ELENgth:DATA 1416
[SENSe<Ch>:]HARMonic:MLENgth:DATA 1416
[SENSe<Ch>:]HARMonic:PERMittivity:DATA 1416
[SENSe<Ch>:]HARMonic:RTIMe:DATA 1417
[SENSe<Ch>:]HARMonic:RTIMe:THReshold 1417
[SENSe<Ch>:]HARMonic:VELocity:DATA 1417
[SENSe<Ch>:]HARMonic? 1415
[SENSe<Ch>:]LPORt<LogPt>:ZCOMmon 1418
[SENSe<Ch>:]LPORt<LogPt>:ZDEFault[:STATe] 1419
[SENSe<Ch>:]LPORt<LogPt>:ZDIFferent 1418
[SENSe<Ch>:]NFIGure:CALibration:GTIMe 1420
[SENSe<Ch>:]NFIGure:CALibration:NTIMe 1420
[SENSe<Ch>:]NFIGure:DEFine 1420
[SENSe<Ch>:]NFIGure:GTIMe 1421
[SENSe<Ch>:]NFIGure:HARMonic[:MAXFrequency] 1421
[SENSe<Ch>:]NFIGure:HARMonic[:MAXimumorder] 1421
[SENSe<Ch>:]NFIGure:HARMonic[:MINFrequency] 1421
[SENSe<Ch>:]NFIGure:HARMonic[:STATe] 1422
[SENSe<Ch>:]NFIGure:IFConst 1422
[SENSe<Ch>:]NFIGure:IFConst:STATe 1422
[SENSe<Ch>:]NFIGure:NTIMe 1423
[SENSe<Ch>:]NFIGure:TEATtenuator 1423
[SENSe<Ch>:]NFIGure:VIRejection[:STATe] 1423
[SENSe<Ch>:]PAE:DCINput:MAIN 1424

[SENSe<Ch>:]PAE:DCINput:SECondary 1424
[SENSe<Ch>:]PAE:PARameters:I 1424
[SENSe<Ch>:]PAE:PARameters:R 1425
[SENSe<Ch>:]PAE:PARameters:U 1425
[SENSe<Ch>:]PAE:TYPE 1425
[SENSe<Ch>:]PORT<PhyPt>:ZREFerence 1481
[SENSe<Ch>:]POWer:AGCMode:ACQuire 1427
[SENSe<Ch>:]POWer:AGCMode:SAVE 1428
[SENSe<Ch>:]POWer:AGCMode<PhyPt>:MEASure 1427
[SENSe<Ch>:]POWer:ATTenuation 1428
[SENSe<Ch>:]POWer:GAINcontrol 1429
[SENSe<Ch>:]POWer:GAINcontrol:ALL 1429
[SENSe<Ch>:]POWer:GAINcontrol:GLOBal 1430
[SENSe<Ch>:]POWer:IFGain<PhyPt>:MEASure 1427
[SENSe<Ch>:]POWer:LOSuppress[:STATe] 1431
[SENSe<Ch>:]SEGMent:COUNt? 1436
[SENSe<Ch>:]SEGMent:POWer[:LEVel]:CONTrol 1449
[SENSe<Ch>:]SEGMent<Seg>:ADD 1433
[SENSe<Ch>:]SEGMent<Seg>:BWIDth[:RESolution] 1434
[SENSe<Ch>:]SEGMent<Seg>:BWIDth[:RESolution]:CONTrol 1434
[SENSe<Ch>:]SEGMent<Seg>:BWIDth[:RESolution]:SELect 1435
[SENSe<Ch>:]SEGMent<Seg>:BWIDth[:RESolution]:SELect:CONTrol 1435
[SENSe<Ch>:]SEGMent<Seg>:CLEar 1641
[SENSe<Ch>:]SEGMent<Seg>:DEFine 1437
[SENSe<Ch>:]SEGMent<Seg>:DEFine:SELect 1438
[SENSe<Ch>:]SEGMent<Seg>:DELete:ALL 1439
[SENSe<Ch>:]SEGMent<Seg>:DELete[:DUMMy] 1439
[SENSe<Ch>:]SEGMent<Seg>:FREQuency:CENTer? 1440
[SENSe<Ch>:]SEGMent<Seg>:FREQuency:SPAN? 1440
[SENSe<Ch>:]SEGMent<Seg>:FREQuency:STARt 1440
[SENSe<Ch>:]SEGMent<Seg>:FREQuency:STEP 1441
[SENSe<Ch>:]SEGMent<Seg>:FREQuency:STOP 1440
[SENSe<Ch>:]SEGMent<Seg>:INSert 1442
[SENSe<Ch>:]SEGMent<Seg>:INSert:SELect 1444
[SENSe<Ch>:]SEGMent<Seg>:LO 1441
[SENSe<Ch>:]SEGMent<Seg>:NAME 1441
[SENSe<Ch>:]SEGMent<Seg>:OVERlap 1641
[SENSe<Ch>:]SEGMent<Seg>:PORT<PortId>:POWer[:LEVel] 1444
[SENSe<Ch>:]SEGMent<Seg>:PORT<PortId>[:STATe] 1445
[SENSe<Ch>:]SEGMent<Seg>:POWer:GAINcontrol 1445
[SENSe<Ch>:]SEGMent<Seg>:POWer:GAINcontrol:ALL 1447
[SENSe<Ch>:]SEGMent<Seg>:POWer:GAINcontrol:CONTrol 1448
[SENSe<Ch>:]SEGMent<Seg>:POWer[:LEVel] 1448
[SENSe<Ch>:]SEGMent<Seg>:SWEep:DETector:TIME 1450
[SENSe<Ch>:]SEGMent<Seg>:SWEep:DETector:TIME:CONTrol 1450
[SENSe<Ch>:]SEGMent<Seg>:SWEep:DWELl 1450
[SENSe<Ch>:]SEGMent<Seg>:SWEep:DWELl:CONTrol 1451
[SENSe<Ch>:]SEGMent<Seg>:SWEep:GENeration 1452
[SENSe<Ch>:]SEGMent<Seg>:SWEep:POINts 1453
[SENSe<Ch>:]SEGMent<Seg>:SWEep:TIME 1453

[SENSe<Ch>:]SEGMent<Seg>:SWEep:TIME:CONTrol ... 1454
[SENSe<Ch>:]SEGMent<Seg>:SWEep:TIME:SUM? ... 1455
[SENSe<Ch>:]SEGMent<Seg>[:STATe] ... 1450
[SENSe<Ch>:]SWEep:AXIS:FREQuency ... 1469
[SENSe<Ch>:]SWEep:AXIS:POWer ... 1470
[SENSe<Ch>:]SWEep:COUNt ... 1471
[SENSe<Ch>:]SWEep:DETector:TIME ... 1472
[SENSe<Ch>:]SWEep:DWELl ... 1472
[SENSe<Ch>:]SWEep:DWELl:IPOint ... 1473
[SENSe<Ch>:]SWEep:GENeration ... 1473
[SENSe<Ch>:]SWEep:GENeration:ANALog:CONDition? ... 1474
[SENSe<Ch>:]SWEep:MODE ... 1475
[SENSe<Ch>:]SWEep:POINts ... 1475
[SENSe<Ch>:]SWEep:SPACing ... 1642
[SENSe<Ch>:]SWEep:STEP ... 1476
[SENSe<Ch>:]SWEep:TIME ... 1477
[SENSe<Ch>:]SWEep:TIME:AUTO ... 1478
[SENSe<Ch>:]SWEep:TYPE ... 1479
[SENSe<Chn>:]CORRection:NSTate? ... 1388
[SENSe<Chn>:]CORRection:PSTate? ... 1395
[SENSe<Chn>:]CORRection:SSTate? ... 1397
[SENSe<Chn>:]FUNCtion[:ON] ... 1642
[SENSe<Chn>:]SWEep:SRCPort ... 1476
*CLS ... 913
*ESE ... 913
*ESR? ... 913
*IDN? ... 914
*IST? ... 914
*OPC ... 914
*OPT? ... 914
*PCB ... 915
*PRE ... 915
*PSC ... 915
*RST ... 915
*SRE ... 915
*STB? ... 916
*TRG ... 916
*TST? ... 916
*WAI ... 916
CALCulate:CALValidate:AVERage[:STATe] ... 918
CALCulate:CALValidate:FORMat ... 918
CALCulate:CALValidate:MODE ... 918
CALCulate:CALValidate:RESPonse:LMAGnitude:LOWer ... 919
CALCulate:CALValidate:RESPonse:LMAGnitude:UPPer ... 919
CALCulate:CALValidate:RESPonse:MAGNitude:LOWer ... 919
CALCulate:CALValidate:RESPonse:MAGNitude:UPPer ... 919
CALCulate:CALValidate:RESPonse:PHASe:LOWer ... 919
CALCulate:CALValidate:RESPonse:PHASe:UPPer ... 919
CALCulate:CALValidate:RESPonse:PRESet ... 919
CALCulate:CALValidate:RUN ... 920

CALCulate:CALValidate:RUN:RESult? 920
CALCulate:CALValidate:STANdard 920
CALCulate:CLIMits:FAIL? 1168
CALCulate:CLIMits[:STATe] 1168
CALCulate:DATA:ALL? 924
CALCulate:DATA:DALL? 926
CALCulate:DATA:TRACe 929
CALCulate:FILTer[:GATE]:TIME:AOFFset 965
CALCulate:FMODel:COPY:CHANnel:DEEMbedding:BALanced 1013
CALCulate:FMODel:COPY:CHANnel:DEEMbedding:PSET 1013
CALCulate:FMODel:COPY:CHANnel:DEEMbedding:SINGleended 1013
CALCulate:FMODel:DCAL:APPLy 971
CALCulate:FMODel:DCAL:CREate:CHANnel 972
CALCulate:FMODel:DCAL:REQuest:CHANnel? 972
CALCulate:FMODel:DEASsistant:COUPon:LEFT:CLEar 972
CALCulate:FMODel:DEASsistant:COUPon:LEFT:FILE 972
CALCulate:FMODel:DEASsistant:COUPon:LEFT:MEASure 973
CALCulate:FMODel:DEASsistant:COUPon:LEFT:PORT 973
CALCulate:FMODel:DEASsistant:COUPon:LEFT:TYPE 973
CALCulate:FMODel:DEASsistant:COUPon:RIGHt:CLEar 972
CALCulate:FMODel:DEASsistant:COUPon:RIGHt:FILE 972
CALCulate:FMODel:DEASsistant:COUPon:RIGHt:MEASure 973
CALCulate:FMODel:DEASsistant:COUPon:RIGHt:PORT 973
CALCulate:FMODel:DEASsistant:COUPon:RIGHt:TYPE 973
CALCulate:FMODel:DEASsistant:DUT:CLEar 974
CALCulate:FMODel:DEASsistant:DUT:FILE 974
CALCulate:FMODel:DEASsistant:DUT:MEASure 974
CALCulate:FMODel:DEASsistant:DUT:PORT 975
CALCulate:FMODel:DEASsistant:DUT:TYPE 975
CALCulate:FMODel:DEASsistant:FIXTure:LEFT:TYPE 976
CALCulate:FMODel:DEASsistant:FIXTure:RIGHt:TYPE 976
CALCulate:FMODel:DEASsistant:IMPCorrect 977
CALCulate:FMODel:DEASsistant:PRESet 977
CALCulate:FMODel:DEASsistant:RUN 977
CALCulate:FMODel:DEASsistant:RUN:RESult? 977
CALCulate:FMODel:DEASsistant:SAMCoupon 977
CALCulate:FMODel:DEASsistant:TOOL 978
CALCulate:FMODel:DELT:CUToff:FREQuency 979
CALCulate:FMODel:DELT:CUToff:STATe 979
CALCulate:FMODel:DELT:DIRectory 980
CALCulate:FMODel:DELT:DIRectory:DEFault 980
CALCulate:FMODel:DELT:FITTing:ORDer 980
CALCulate:FMODel:DELT:FREQuencies:CURRent:COUNt? 981
CALCulate:FMODel:DELT:FREQuencies:CURRent? 980
CALCulate:FMODel:DELT:FREQuencies:DEFault:COUNt? 981
CALCulate:FMODel:DELT:FREQuencies:DEFault? 981
CALCulate:FMODel:DELT:FREQuencies:USEDefault 981
CALCulate:FMODel:DELT:FREQuencies:USER:ADD 982
CALCulate:FMODel:DELT:FREQuencies:USER:COUNt? 983
CALCulate:FMODel:DELT:FREQuencies:USER:DELete 982

CALCulate:FMODel:DELT:FREQuencies:USER:DELete:ALL....983
CALCulate:FMODel:DELT:FREQuencies:USER?....982
CALCulate:FMODel:DELT:M1L:CACHe:CLEar:ALL....983
CALCulate:FMODel:DELT:M1L:CACHe:CLEar:SELected....983
CALCulate:FMODel:DELT:M1L:DIFFmode....984
CALCulate:FMODel:DELT:M1L:FILename....984
CALCulate:FMODel:DELT:M1L:LENGth....984
CALCulate:FMODel:DELT:M1L:MEASure....985
CALCulate:FMODel:DELT:M2L:FILename....984
CALCulate:FMODel:DELT:M2L:LENGth....984
CALCulate:FMODel:DELT:M2L:MEASure....985
CALCulate:FMODel:DELT:M3L:FILename....984
CALCulate:FMODel:DELT:M3L:LENGth....984
CALCulate:FMODel:DELT:M3L:MEASure....985
CALCulate:FMODel:DELT:MEASurement....985
CALCulate:FMODel:DELT:METHod....986
CALCulate:FMODel:DELT:PORDer....986
CALCulate:FMODel:DELT:RESonance....986
CALCulate:FMODel:DELT:RUN....987
CALCulate:FMODel:DELT:SWEep:CONTrol....987
CALCulate:FMODel:DELT:SWEep:CONTrol:IFBW?....987
CALCulate:FMODel:DELT:SWEep:CONTrol:STEP?....987
CALCulate:FMODel:DELT:SWEep:FREQuency:MAXimum....988
CALCulate:FMODel:DELT:SWEep:FREQuency:MINimum....988
CALCulate:FMODel:DELT:TCONfig....988
CALCulate:FMODel:DELT:TDR....988
CALCulate:FMODel:DELT<Ph_pt>:M1L[:STATe]....985
CALCulate:FMODel:DELT<Ph_pt>:M2L[:STATe]....985
CALCulate:FMODel:DELT<Ph_pt>:M3L[:STATe]....985
CALCulate:FMODel:DIRectory....1013
CALCulate:FMODel:DIRectory:DEFault....1013
CALCulate:FMODel:DIRectory:DEFault:CLEar....1013
CALCulate:FMODel:EZD:COUPon:MEASure....990
CALCulate:FMODel:EZD:COUPon:MEASure:FILename....990
CALCulate:FMODel:EZD:COUPon:MEASure:FILename:CLEar....990
CALCulate:FMODel:EZD:DCEXtrapolat....991
CALCulate:FMODel:EZD:DEPLane:STATe....991
CALCulate:FMODel:EZD:DEPLane<Pl>....991
CALCulate:FMODel:EZD:DUT:MEASure....992
CALCulate:FMODel:EZD:DUT:MEASure:FILename....992
CALCulate:FMODel:EZD:DUT:MEASure:FILename:CLEar....992
CALCulate:FMODel:EZD:GENerate:SIDE<1|2>....993
CALCulate:FMODel:EZD:IMPedance....994
CALCulate:FMODel:EZD:IMPedance:BWIDlimit....994
CALCulate:FMODel:EZD:IMPedance:REFerence....994
CALCulate:FMODel:EZD:PORT:ORDer....994
CALCulate:FMODel:EZD:PRESet....995
CALCulate:FMODel:EZD:RUN:RUN....995
CALCulate:FMODel:EZD:VERSion?....996
CALCulate:FMODel:EZD<Ph_pt>:CAUSality....989

CALCulate:FMODel:EZD<Ph_pt>:COUPon[:STATe] 990
CALCulate:FMODel:EZD<Ph_pt>:DELay:LEFT 991
CALCulate:FMODel:EZD<Ph_pt>:DELay:RIGHt 991
CALCulate:FMODel:EZD<Ph_pt>:DUT[:STATe] 992
CALCulate:FMODel:EZD<Ph_pt>:FASTmode 993
CALCulate:FMODel:EZD<Ph_pt>:HFC 993
CALCulate:FMODel:EZD<Ph_pt>:ICALculation 993
CALCulate:FMODel:EZD<Ph_pt>:SURFace 996
CALCulate:FMODel:EZD<Pt>:RUN[:STATe] 995
CALCulate:FMODel:ISD:ATTenuation:BEHavior 997
CALCulate:FMODel:ISD:COUPon:MEASure 997
CALCulate:FMODel:ISD:COUPon:MEASure:FILename 998
CALCulate:FMODel:ISD:COUPon:MEASure:FILename:CLEar 998
CALCulate:FMODel:ISD:COUPon:MEASure:OPEN 998
CALCulate:FMODel:ISD:COUPon:MEASure:OPEN:FILename 998
CALCulate:FMODel:ISD:COUPon:MEASure:SHORt 999
CALCulate:FMODel:ISD:COUPon:MEASure:SHORt:FILename 999
CALCulate:FMODel:ISD:COUPon:TYPE 999
CALCulate:FMODel:ISD:DCEXtrapolat 1000
CALCulate:FMODel:ISD:DIFFonly 1000
CALCulate:FMODel:ISD:DUT:MEASure 1000
CALCulate:FMODel:ISD:DUT:MEASure:FILename 1001
CALCulate:FMODel:ISD:DUT:MEASure:FILename:CLEar 1001
CALCulate:FMODel:ISD:DUT:TYPE 1001
CALCulate:FMODel:ISD:FTIMe:DUT 1002
CALCulate:FMODel:ISD:FTIMe:OVERride 1002
CALCulate:FMODel:ISD:IMPedance 1002
CALCulate:FMODel:ISD:OPERation 1003
CALCulate:FMODel:ISD:PASSivity 1003
CALCulate:FMODel:ISD:PORT:ORDer 1003
CALCulate:FMODel:ISD:PORT:SKIP 1004
CALCulate:FMODel:ISD:PORT:SKIP:LEFT 1004
CALCulate:FMODel:ISD:PORT:SKIP:NONE 1004
CALCulate:FMODel:ISD:PORT:SKIP:RIGHt 1004
CALCulate:FMODel:ISD:PRESet 1004
CALCulate:FMODel:ISD:RUN:RUN 1004
CALCulate:FMODel:ISD:SCALe:ATTenuation 1005
CALCulate:FMODel:ISD:SCALe:FREQuency 1005
CALCulate:FMODel:ISD:SCALe:FTIMe 1006
CALCulate:FMODel:ISD:SCALe:FTIMe:TYPE 1006
CALCulate:FMODel:ISD:SMALlfixture 1006
CALCulate:FMODel:ISD:SMOThing 1007
CALCulate:FMODel:ISD:TRACe:COUPling 1007
CALCulate:FMODel:ISD:VERSion? 1007
CALCulate:FMODel:ISD<Ph_pt>:COUPon[:STATe] 999
CALCulate:FMODel:ISD<Ph_pt>:DUT[:STATe] 1001
CALCulate:FMODel:ISD<Pt>:RUN[:STATe] 1005
CALCulate:FMODel:QSETup 1013
CALCulate:FMODel:REName 1014
CALCulate:FMODel:SFD:AUTO 1008

CALCulate:FMODel:SFD:COUPon:MEASure....1008
CALCulate:FMODel:SFD:COUPon:MEASure:FILename....1008
CALCulate:FMODel:SFD:COUPon:MEASure:FILename:CLEar....1009
CALCulate:FMODel:SFD:COUPon:TYPE....1009
CALCulate:FMODel:SFD:DIFFcfg....1009
CALCulate:FMODel:SFD:DUT:MEASure....1010
CALCulate:FMODel:SFD:DUT:MEASure:FILename....1010
CALCulate:FMODel:SFD:DUT:MEASure:FILename:CLEar....1010
CALCulate:FMODel:SFD:IMPedance....1011
CALCulate:FMODel:SFD:PRESet....1011
CALCulate:FMODel:SFD:RUN:RUN....1011
CALCulate:FMODel:SFD:TOTaldiffcfg....1012
CALCulate:FMODel:SFD:VERSion?....1012
CALCulate:FMODel:SFD<Ph_pt>:COUPon[:STATe]....1009
CALCulate:FMODel:SFD<Ph_pt>:DUT[:STATe]....1010
CALCulate:FMODel:SFD<Ph_pt>:RUN[:STATe]....1012
CALCulate:LIMit:CIRCle:FAIL:ALL?....1016
CALCulate:LIMit:FAIL:ALL?....1022
CALCulate:LIMit:FAIL:DATA?....1022
CALCulate:LIMit:POINts:LOWer?....1025
CALCulate:LIMit:POINts:UPPer?....1025
CALCulate:MARKer:COUPled:TYPE....1036
CALCulate:MARKer:COUPled[:STATe]....1035
CALCulate:MARKer:FUNCtion:BWIDth:GMCenter....1040
CALCulate:PARameter:DELete:ALL....1068
CALCulate:PARameter:DELete:MEMory....1068
CALCulate:PARameter:MOVE....1070
CALCulate:PARameter:NEW:CHANnel....1070
CALCulate:PIPelining:DATA?....1172
CALCulate:RIPPle:DISPlay:RESult:ALL[:STATe]....1084
CALCulate:RIPPle:FAIL:ALL?....1085
CALCulate:TDVSwr[:STATe]....1174
CALCulate:TRANsform:DTFault:EXPort....1101
CALCulate<Ch>:DATA:CALL....924
CALCulate<Ch>:DATA:CALL:CATalog?....925
CALCulate<Ch>:DATA:CHANnel:ALL?....925
CALCulate<Ch>:DATA:CHANnel:DALL?....926
CALCulate<Ch>:DATA:MDATa:INTerpolate....926
CALCulate<Ch>:DATA:SGRoup?....929
CALCulate<Ch>:PARameter:CATalog:SENDed?....1064
CALCulate<Ch>:PARameter:CATalog?....1064
CALCulate<Ch>:PARameter:COPY....1064
CALCulate<Ch>:PARameter:COPY:CHANnel....1065
CALCulate<Ch>:PARameter:DEFine....1624
CALCulate<Ch>:PARameter:DEFine:SGRoup....1065
CALCulate<Ch>:PARameter:DELete....1067
CALCulate<Ch>:PARameter:DELete:CALL....1068
CALCulate<Ch>:PARameter:DELete:CMEMory....1068
CALCulate<Ch>:PARameter:DELete:SGRoup....1068
CALCulate<Ch>:PARameter:MEASure....1069

CALCulate<Ch>:PARameter:MEASure:SENDed....1069
CALCulate<Ch>:PARameter:SDEFine....1071
CALCulate<Ch>:PARameter:SDEFine:SENDed....1074
CALCulate<Ch>:PARameter:SELect....1074
CALCulate<Ch>:TRANsform:VNETworks:ACTivateall[:STATe]....1122
CALCulate<Ch>:TRANsform:VNETworks:BALanced:DEEMbedding<LogPt>:PARameters:C<Cmp>....1123
CALCulate<Ch>:TRANsform:VNETworks:BALanced:DEEMbedding<LogPt>:PARameters:DATA<Port>....1124
CALCulate<Ch>:TRANsform:VNETworks:BALanced:DEEMbedding<LogPt>:PARameters:G<Cmp>....1125
CALCulate<Ch>:TRANsform:VNETworks:BALanced:DEEMbedding<LogPt>:PARameters:L<Cmp>....1125
CALCulate<Ch>:TRANsform:VNETworks:BALanced:DEEMbedding<LogPt>:PARameters:R<Cmp>....1126
CALCulate<Ch>:TRANsform:VNETworks:BALanced:DEEMbedding<LogPt>:TNDefinition....1127
CALCulate<Ch>:TRANsform:VNETworks:BALanced:DEEMbedding<LogPt>[:STATe]....1127
CALCulate<Ch>:TRANsform:VNETworks:BALanced:EMBedding<LogPt>:PARameters:C<Cmp>....1128
CALCulate<Ch>:TRANsform:VNETworks:BALanced:EMBedding<LogPt>:PARameters:DATA<Port>....1129
CALCulate<Ch>:TRANsform:VNETworks:BALanced:EMBedding<LogPt>:PARameters:G<Cmp>....1130
CALCulate<Ch>:TRANsform:VNETworks:BALanced:EMBedding<LogPt>:PARameters:L<Cmp>....1130
CALCulate<Ch>:TRANsform:VNETworks:BALanced:EMBedding<LogPt>:PARameters:R<Cmp>....1131
CALCulate<Ch>:TRANsform:VNETworks:BALanced:EMBedding<LogPt>:TNDefinition....1132
CALCulate<Ch>:TRANsform:VNETworks:BALanced:EMBedding<LogPt>[:STATe]....1132
CALCulate<Ch>:TRANsform:VNETworks:DIFFerential:EMBedding<LogPt>:PARameters:C<Cmp>....1133
CALCulate<Ch>:TRANsform:VNETworks:DIFFerential:EMBedding<LogPt>:PARameters:DATA....1133
CALCulate<Ch>:TRANsform:VNETworks:DIFFerential:EMBedding<LogPt>:PARameters:G<Cmp>....1134
CALCulate<Ch>:TRANsform:VNETworks:DIFFerential:EMBedding<LogPt>:PARameters:L<Cmp>....1134
CALCulate<Ch>:TRANsform:VNETworks:DIFFerential:EMBedding<LogPt>:PARameters:R<Cmp>....1135
CALCulate<Ch>:TRANsform:VNETworks:DIFFerential:EMBedding<LogPt>:TNDefinition....1136
CALCulate<Ch>:TRANsform:VNETworks:DIFFerential:EMBedding<LogPt>[:STATe]....1135
CALCulate<Ch>:TRANsform:VNETworks:FSIMulator[:STATe]....1136
CALCulate<Ch>:TRANsform:VNETworks:GLOop:DEEMbedding<group>:PARameters:C....1136
CALCulate<Ch>:TRANsform:VNETworks:GLOop:DEEMbedding<group>:PARameters:G....1137
CALCulate<Ch>:TRANsform:VNETworks:GLOop:DEEMbedding<group>:PARameters:L....1138
CALCulate<Ch>:TRANsform:VNETworks:GLOop:DEEMbedding<group>:PARameters:R....1138
CALCulate<Ch>:TRANsform:VNETworks:GLOop:DEEMbedding<group>:TNDefinition....1140
CALCulate<Ch>:TRANsform:VNETworks:GLOop:DEEMbedding<group>[:STATe]....1139
CALCulate<Ch>:TRANsform:VNETworks:GLOop:EMBedding<group>:PARameters:C....1140
CALCulate<Ch>:TRANsform:VNETworks:GLOop:EMBedding<group>:PARameters:G....1141
CALCulate<Ch>:TRANsform:VNETworks:GLOop:EMBedding<group>:PARameters:L....1141
CALCulate<Ch>:TRANsform:VNETworks:GLOop:EMBedding<group>:PARameters:R....1142
CALCulate<Ch>:TRANsform:VNETworks:GLOop:EMBedding<group>:TNDefinition....1143
CALCulate<Ch>:TRANsform:VNETworks:GLOop:EMBedding<group>[:STATe]....1143
CALCulate<Ch>:TRANsform:VNETworks:GLOop:GROup....1144
CALCulate<Ch>:TRANsform:VNETworks:PPAir:DEEMbedding:DELete....1145
CALCulate<Ch>:TRANsform:VNETworks:PPAir:DEEMbedding<ListId>:DEFine....1144
CALCulate<Ch>:TRANsform:VNETworks:PPAir:DEEMbedding<ListId>:PARameters:C<1|2|3>....1145
CALCulate<Ch>:TRANsform:VNETworks:PPAir:DEEMbedding<ListId>:PARameters:G<1|2|3>....1146
CALCulate<Ch>:TRANsform:VNETworks:PPAir:DEEMbedding<ListId>:PARameters:L<1|2|3>....1147
CALCulate<Ch>:TRANsform:VNETworks:PPAir:DEEMbedding<ListId>:PARameters:R<1|2|3>....1147
CALCulate<Ch>:TRANsform:VNETworks:PPAir:DEEMbedding<ListId>:TNDefinition....1149
CALCulate<Ch>:TRANsform:VNETworks:PPAir:DEEMbedding<ListId>[:STATe]....1148
CALCulate<Ch>:TRANsform:VNETworks:PPAir:EMBedding<ListId>:DEFine....1149
CALCulate<Ch>:TRANsform:VNETworks:PPAir:EMBedding<ListId>:DELete....1150

CALCulate<Ch>:TRANsform:VNETworks:PPAir:EMBedding<ListId>:PARameters:C<1|2|3>..... 1150
CALCulate<Ch>:TRANsform:VNETworks:PPAir:EMBedding<ListId>:PARameters:G<1|2|3>..... 1152
CALCulate<Ch>:TRANsform:VNETworks:PPAir:EMBedding<ListId>:PARameters:L<1|2|3>..... 1151
CALCulate<Ch>:TRANsform:VNETworks:PPAir:EMBedding<ListId>:PARameters:R<1|2|3>..... 1151
CALCulate<Ch>:TRANsform:VNETworks:PPAir:EMBedding<ListId>:TNDefinition..... 1153
CALCulate<Ch>:TRANsform:VNETworks:PPAir:EMBedding<ListId>[:STATe]..... 1153
CALCulate<Ch>:TRANsform:VNETworks:PSET:DEEMbedding<ListId>:DEFine..... 1154
CALCulate<Ch>:TRANsform:VNETworks:PSET:EMBedding<ListId>:DEFine..... 1154
CALCulate<Ch>:TRANsform:VNETworks:SENDed:DEEMbedding<PhyPt>:IGNore..... 1154
CALCulate<Ch>:TRANsform:VNETworks:SENDed:DEEMbedding<PhyPt>:PARameters:C<Cmp>..... 1155
CALCulate<Ch>:TRANsform:VNETworks:SENDed:DEEMbedding<PhyPt>:PARameters:DATA..... 1155
CALCulate<Ch>:TRANsform:VNETworks:SENDed:DEEMbedding<PhyPt>:PARameters:G<Cmp>..... 1156
CALCulate<Ch>:TRANsform:VNETworks:SENDed:DEEMbedding<PhyPt>:PARameters:L<Cmp>..... 1157
CALCulate<Ch>:TRANsform:VNETworks:SENDed:DEEMbedding<PhyPt>:PARameters:R<Cmp>..... 1158
CALCulate<Ch>:TRANsform:VNETworks:SENDed:DEEMbedding<PhyPt>:TNDefinition..... 1159
CALCulate<Ch>:TRANsform:VNETworks:SENDed:DEEMbedding<PhyPt>[:STATe]..... 1158
CALCulate<Ch>:TRANsform:VNETworks:SENDed:EMBedding<PhyPt>:PARameters:C<Cmp>..... 1159
CALCulate<Ch>:TRANsform:VNETworks:SENDed:EMBedding<PhyPt>:PARameters:DATA..... 1160
CALCulate<Ch>:TRANsform:VNETworks:SENDed:EMBedding<PhyPt>:PARameters:G<Cmp>..... 1161
CALCulate<Ch>:TRANsform:VNETworks:SENDed:EMBedding<PhyPt>:PARameters:L<Cmp>..... 1161
CALCulate<Ch>:TRANsform:VNETworks:SENDed:EMBedding<PhyPt>:PARameters:R<Cmp>..... 1162
CALCulate<Ch>:TRANsform:VNETworks:SENDed:EMBedding<PhyPt>:TNDefinition..... 1163
CALCulate<Ch>:TRANsform:VNETworks:SENDed:EMBedding<PhyPt>[:STATe]..... 1163
CALCulate<Ch>:TRANsform:VNETworks:WAVes..... 1164
CALCulate<Chn>:DATA..... 921
CALCulate<Chn>:DATA:NSWeep:COUNt?..... 927
CALCulate<Chn>:DATA:NSWeep:FIRSt?..... 927
CALCulate<Chn>:DATA:NSWeep[:LAST]?..... 928
CALCulate<Chn>:DATA:STIMulus?..... 929
CALCulate<Chn>:DLINe..... 1168
CALCulate<Chn>:DLINe:STATe..... 1169
CALCulate<Chn>:DTIMe:DATA?..... 930
CALCulate<Chn>:DTIMe:LIMit:FAIL:BEEP..... 931
CALCulate<Chn>:DTIMe:LIMit:FAIL?..... 931
CALCulate<Chn>:DTIMe:LIMit:LIMit..... 932
CALCulate<Chn>:DTIMe:LIMit:STATe..... 932
CALCulate<Chn>:DTIMe:POSition..... 932
CALCulate<Chn>:DTIMe:STATe..... 932
CALCulate<Chn>:DTIMe:TARGet..... 933
CALCulate<Chn>:EYE:DUT:MODE..... 935
CALCulate<Chn>:EYE:EMPHasis:CURSor:POST<1|2>..... 935
CALCulate<Chn>:EYE:EMPHasis:CURSor:PRE..... 936
CALCulate<Chn>:EYE:EMPHasis:STATe..... 936
CALCulate<Chn>:EYE:EQUalization:CTLE:COUNt:POLE..... 936
CALCulate<Chn>:EYE:EQUalization:CTLE:COUNt:ZERO..... 937
CALCulate<Chn>:EYE:EQUalization:CTLE:DC..... 937
CALCulate<Chn>:EYE:EQUalization:CTLE:LIST:POLE..... 937
CALCulate<Chn>:EYE:EQUalization:CTLE:LIST:ZERO..... 938
CALCulate<Chn>:EYE:EQUalization:CTLE:POLE<Num>..... 938
CALCulate<Chn>:EYE:EQUalization:CTLE:RATio..... 938

CALCulate<Chn>:EYE:EQUalization:CTLE:STATe....939
CALCulate<Chn>:EYE:EQUalization:CTLE:ZERO<Num>....939
CALCulate<Chn>:EYE:EQUalization:DFE:COUNt:TAPS....939
CALCulate<Chn>:EYE:EQUalization:DFE:LIST:TAPS....940
CALCulate<Chn>:EYE:EQUalization:DFE:MODE....940
CALCulate<Chn>:EYE:EQUalization:DFE:STATe....940
CALCulate<Chn>:EYE:EQUalization:DFE:TAP<Num>....940
CALCulate<Chn>:EYE:EQUalization:FFE:COUNt:POSTcursor....941
CALCulate<Chn>:EYE:EQUalization:FFE:COUNt:PRECursor....941
CALCulate<Chn>:EYE:EQUalization:FFE:LIST:POSTcursor....941
CALCulate<Chn>:EYE:EQUalization:FFE:LIST:PRECursor....941
CALCulate<Chn>:EYE:EQUalization:FFE:MAIN:TAP....942
CALCulate<Chn>:EYE:EQUalization:FFE:MODE....942
CALCulate<Chn>:EYE:EQUalization:FFE:POSTcursor<Num>....942
CALCulate<Chn>:EYE:EQUalization:FFE:PRECursor<Num>....943
CALCulate<Chn>:EYE:EQUalization:FFE:STATe....943
CALCulate<Chn>:EYE:EQUalization:STATe....943
CALCulate<Chn>:EYE:INPut:BPATtern:TYPE....944
CALCulate<Chn>:EYE:INPut:DRATe....944
CALCulate<Chn>:EYE:INPut:LENGth:BITS....944
CALCulate<Chn>:EYE:INPut:LENGth:PRBS....945
CALCulate<Chn>:EYE:INPut:MODulation....945
CALCulate<Chn>:EYE:INPut:OLEVel....946
CALCulate<Chn>:EYE:INPut:RTIMe:DATA....946
CALCulate<Chn>:EYE:INPut:RTIMe:THReshold....947
CALCulate<Chn>:EYE:INPut:XRATe....946
CALCulate<Chn>:EYE:INPut:ZLEVel....946
CALCulate<Chn>:EYE:JITTer:DIRac:DELTa....947
CALCulate<Chn>:EYE:JITTer:DIRac:PROBability....947
CALCulate<Chn>:EYE:JITTer:PERiodic:FREQuency....948
CALCulate<Chn>:EYE:JITTer:PERiodic:MAGNitude....948
CALCulate<Chn>:EYE:JITTer:PERiodic:PHASe....949
CALCulate<Chn>:EYE:JITTer:RANDom:STDDeviation....949
CALCulate<Chn>:EYE:JITTer:STATe....949
CALCulate<Chn>:EYE:JITTer:TYPE:DIRac....950
CALCulate<Chn>:EYE:JITTer:TYPE:PERiodic....950
CALCulate<Chn>:EYE:JITTer:TYPE:RANDom....951
CALCulate<Chn>:EYE:JITTer:TYPE:USER....951
CALCulate<Chn>:EYE:MASK AUTO....952
CALCulate<Chn>:EYE:MASK:CENTer:HORizontal....952
CALCulate<Chn>:EYE:MASK:CENTer:VERTical....952
CALCulate<Chn>:EYE:MASK:DATA?....952
CALCulate<Chn>:EYE:MASK:FAIL:BEEP....954
CALCulate<Chn>:EYE:MASK:FAIL:CONDition....954
CALCulate<Chn>:EYE:MASK:FAIL?....953
CALCulate<Chn>:EYE:MASK:SHAPe:BOTTom:HORizontal....955
CALCulate<Chn>:EYE:MASK:SHAPe:BOTTom:STATe....955
CALCulate<Chn>:EYE:MASK:SHAPe:BOTTom:VERTical....955
CALCulate<Chn>:EYE:MASK:SHAPe:POLYgon:HORizontal....956
CALCulate<Chn>:EYE:MASK:SHAPe:POLYgon:STATe....955

CALCulate<Chn>:EYE:MASK:SHAPe:POLYgon:TYPE....957
CALCulate<Chn>:EYE:MASK:SHAPe:POLYgon:VERTical....957
CALCulate<Chn>:EYE:MASK:SHAPe:TOP:HORizontal....955
CALCulate<Chn>:EYE:MASK:SHAPe:TOP:STATe....955
CALCulate<Chn>:EYE:MASK:SHAPe:TOP:VERTical....955
CALCulate<Chn>:EYE:MASK:SHOW....958
CALCulate<Chn>:EYE:MASK:STATe....958
CALCulate<Chn>:EYE:MASK:VIOLation:RATE....958
CALCulate<Chn>:EYE:MASK:VIOLation:TOLerance....959
CALCulate<Chn>:EYE:MEASurement:BASic....956
CALCulate<Chn>:EYE:MEASurement:DATA?....959
CALCulate<Chn>:EYE:MEASurement:STATe....960
CALCulate<Chn>:EYE:MEASurement:TIME....956
CALCulate<Chn>:EYE:MEASurement:TTIMe:THReshold....961
CALCulate<Chn>:EYE:NOISe:RMS....961
CALCulate<Chn>:EYE:NOISe:STATe....961
CALCulate<Chn>:EYE:STATe....962
CALCulate<Chn>:EYE:STIMulus:ENCoder....962
CALCulate<Chn>:EYE:STIMulus:LOWPass....962
CALCulate<Chn>:EYE:STIMulus:SCRambler....963
CALCulate<Chn>:EYE:VIEW....963
CALCulate<Chn>:FILTer[:GATE]:TIME:ADD....964
CALCulate<Chn>:FILTer[:GATE]:TIME:CATalog?....965
CALCulate<Chn>:FILTer[:GATE]:TIME:CENTer....965
CALCulate<Chn>:FILTer[:GATE]:TIME:DCHebyshev....966
CALCulate<Chn>:FILTer[:GATE]:TIME:DELete....966
CALCulate<Chn>:FILTer[:GATE]:TIME:LINK....967
CALCulate<Chn>:FILTer[:GATE]:TIME:SELect....967
CALCulate<Chn>:FILTer[:GATE]:TIME:SHAPe....967
CALCulate<Chn>:FILTer[:GATE]:TIME:SHOW....968
CALCulate<Chn>:FILTer[:GATE]:TIME:SPAN....968
CALCulate<Chn>:FILTer[:GATE]:TIME:STARt....969
CALCulate<Chn>:FILTer[:GATE]:TIME:STATe....969
CALCulate<Chn>:FILTer[:GATE]:TIME:STOP....969
CALCulate<Chn>:FILTer[:GATE]:TIME:WINDow....970
CALCulate<Chn>:FILTer[:GATE]:TIME[:TYPE]....970
CALCulate<Chn>:FORMat....1169
CALCulate<Chn>:FORMat:WQUType....1171
CALCulate<Chn>:GDAPerture:SCOunt....1171
CALCulate<Chn>:IAVerage:MODE....1171
CALCulate<Chn>:IAVerage[:STATe]....1172
CALCulate<Chn>:LIMit:CIRCle:CLEar....1015
CALCulate<Chn>:LIMit:CIRCle:DATA....1015
CALCulate<Chn>:LIMit:CIRCle:DISPlay[:STATe]....1015
CALCulate<Chn>:LIMit:CIRCle:FAIL?....1016
CALCulate<Chn>:LIMit:CIRCle:SOUNd[:STATe]....1016
CALCulate<Chn>:LIMit:CIRCle[:STATe]....1017
CALCulate<Chn>:LIMit:CLEar....1017
CALCulate<Chn>:LIMit:CONTrol:DOMain....1621
CALCulate<Chn>:LIMit:CONTrol:SHIFt....1018

CALCulate<Chn>:LIMit:CONTrol[:DATA] 1017
CALCulate<Chn>:LIMit:DATA 1019
CALCulate<Chn>:LIMit:DCIRcle:CLEar 1020
CALCulate<Chn>:LIMit:DCIRcle:DATA 1020
CALCulate<Chn>:LIMit:DCIRcle:DISPlay[:STATe] 1020
CALCulate<Chn>:LIMit:DCIRcle[:STATe] 1020
CALCulate<Chn>:LIMit:DELete:ALL 1021
CALCulate<Chn>:LIMit:DISPlay[:STATe] 1021
CALCulate<Chn>:LIMit:FAIL? 1021
CALCulate<Chn>:LIMit:LOWer:FEED 1024
CALCulate<Chn>:LIMit:LOWer:SHIFt 1025
CALCulate<Chn>:LIMit:LOWer:STATe 1623
CALCulate<Chn>:LIMit:LOWer[:DATA] 1022
CALCulate<Chn>:LIMit:RDOMain:COMPlex 1622
CALCulate<Chn>:LIMit:RDOMain:FORMat 1623
CALCulate<Chn>:LIMit:RDOMain:SPACing 1623
CALCulate<Chn>:LIMit:SEGMent:COUNt? 1027
CALCulate<Chn>:LIMit:SEGMent<Seg>:AMPLitude:STARt 1026
CALCulate<Chn>:LIMit:SEGMent<Seg>:AMPLitude:STOP 1026
CALCulate<Chn>:LIMit:SEGMent<Seg>:FORMula 1027
CALCulate<Chn>:LIMit:SEGMent<Seg>:FORMula:STATe 1028
CALCulate<Chn>:LIMit:SEGMent<Seg>:INTerpol 1028
CALCulate<Chn>:LIMit:SEGMent<Seg>:STIMulus:STARt 1029
CALCulate<Chn>:LIMit:SEGMent<Seg>:STIMulus:STOP 1029
CALCulate<Chn>:LIMit:SEGMent<Seg>:TYPE 1029
CALCulate<Chn>:LIMit:SOUNd[:STATe] 1030
CALCulate<Chn>:LIMit:STATe 1030
CALCulate<Chn>:LIMit:STATe:AREA 1031
CALCulate<Chn>:LIMit:TTLout<Pt>[:STATe] 1031
CALCulate<Chn>:LIMit:UPPer:FEED 1024
CALCulate<Chn>:LIMit:UPPer:SHIFt 1025
CALCulate<Chn>:LIMit:UPPer:STATe 1623
CALCulate<Chn>:LIMit:UPPer[:DATA] 1022
CALCulate<Chn>:LIMit:X:OFFSet 1032
CALCulate<Chn>:LIMit:Y:OFFSet 1032
CALCulate<Chn>:MARKer:DEFault:FORMat 1036
CALCulate<Chn>:MARKer:MPEak:EXCursion 1048
CALCulate<Chn>:MARKer:MPEak:EXCursion:STATe 1048
CALCulate<Chn>:MARKer:MPEak:THReshold 1049
CALCulate<Chn>:MARKer:MPEak:THReshold:STATe 1049
CALCulate<Chn>:MARKer:SEARch:BFILter:RESult[:STATe] 1050
CALCulate<Chn>:MARKer:SEARch:BFILter:RESult[:STATe]:AREA 1050
CALCulate<Chn>:MARKer[:STATe]:AREA 1053
CALCulate<Chn>:MARKer<Mk>:AOFF 1034
CALCulate<Chn>:MARKer<Mk>:BWIDth 1034
CALCulate<Chn>:MARKer<Mk>:DELTa[:STATe] 1038
CALCulate<Chn>:MARKer<Mk>:EXCursion 1038
CALCulate<Chn>:MARKer<Mk>:EXCursion:STATe 1039
CALCulate<Chn>:MARKer<Mk>:FORMat 1039
CALCulate<Chn>:MARKer<Mk>:FUNCtion:BWIDth 1625

CALCulate<Chn>:MARKer<Mk>:FUNCtion:BWIDth:MODE....1040
CALCulate<Chn>:MARKer<Mk>:FUNCtion:CENTer....1041
CALCulate<Chn>:MARKer<Mk>:FUNCtion:DELTa:STATe....1626
CALCulate<Chn>:MARKer<Mk>:FUNCtion:DOMain:USER:SHOW....1042
CALCulate<Chn>:MARKer<Mk>:FUNCtion:DOMain:USER:STARt....1042
CALCulate<Chn>:MARKer<Mk>:FUNCtion:DOMain:USER:STOP....1042
CALCulate<Chn>:MARKer<Mk>:FUNCtion:DOMain:USER[:RANGe]....1041
CALCulate<Chn>:MARKer<Mk>:FUNCtion:EXECute....1043
CALCulate<Chn>:MARKer<Mk>:FUNCtion:MAX....1044
CALCulate<Chn>:MARKer<Mk>:FUNCtion:MIN....1044
CALCulate<Chn>:MARKer<Mk>:FUNCtion:PEAK....1044
CALCulate<Chn>:MARKer<Mk>:FUNCtion:REFVal....1045
CALCulate<Chn>:MARKer<Mk>:FUNCtion:RESult?....1045
CALCulate<Chn>:MARKer<Mk>:FUNCtion:SETRef....1045
CALCulate<Chn>:MARKer<Mk>:FUNCtion:SPAN....1046
CALCulate<Chn>:MARKer<Mk>:FUNCtion:STARt....1046
CALCulate<Chn>:MARKer<Mk>:FUNCtion:STOP....1046
CALCulate<Chn>:MARKer<Mk>:FUNCtion:TARGet....1627
CALCulate<Chn>:MARKer<Mk>:FUNCtion:ZERodelay....1047
CALCulate<Chn>:MARKer<Mk>:FUNCtion[:SELect]....1626
CALCulate<Chn>:MARKer<Mk>:INFO....1047
CALCulate<Chn>:MARKer<Mk>:MAXimum....1627
CALCulate<Chn>:MARKer<Mk>:MINimum....1627
CALCulate<Chn>:MARKer<Mk>:MODE....1047
CALCulate<Chn>:MARKer<Mk>:NAME....1049
CALCulate<Chn>:MARKer<Mk>:REFerence:EXCursion....1038
CALCulate<Chn>:MARKer<Mk>:REFerence:EXCursion:STATe....1039
CALCulate<Chn>:MARKer<Mk>:REFerence:FORMat....1039
CALCulate<Chn>:MARKer<Mk>:REFerence:FUNCtion:CENTer....1041
CALCulate<Chn>:MARKer<Mk>:REFerence:FUNCtion:DOMain:USER:SHOW....1042
CALCulate<Chn>:MARKer<Mk>:REFerence:FUNCtion:DOMain:USER:STARt....1042
CALCulate<Chn>:MARKer<Mk>:REFerence:FUNCtion:DOMain:USER:STOP....1042
CALCulate<Chn>:MARKer<Mk>:REFerence:FUNCtion:DOMain:USER[:RANGe]....1041
CALCulate<Chn>:MARKer<Mk>:REFerence:FUNCtion:EXECute....1043
CALCulate<Chn>:MARKer<Mk>:REFerence:FUNCtion:MAX....1044
CALCulate<Chn>:MARKer<Mk>:REFerence:FUNCtion:MIN....1044
CALCulate<Chn>:MARKer<Mk>:REFerence:FUNCtion:REFVal....1045
CALCulate<Chn>:MARKer<Mk>:REFerence:FUNCtion:RESult?....1045
CALCulate<Chn>:MARKer<Mk>:REFerence:FUNCtion:STARt....1046
CALCulate<Chn>:MARKer<Mk>:REFerence:FUNCtion:STOP....1046
CALCulate<Chn>:MARKer<Mk>:REFerence:FUNCtion:TARGet....1627
CALCulate<Chn>:MARKer<Mk>:REFerence:FUNCtion:ZERodelay....1047
CALCulate<Chn>:MARKer<Mk>:REFerence:FUNCtion[:SELect]....1626
CALCulate<Chn>:MARKer<Mk>:REFerence:MAXimum....1627
CALCulate<Chn>:MARKer<Mk>:REFerence:MINimum....1627
CALCulate<Chn>:MARKer<Mk>:REFerence:MODE....1047
CALCulate<Chn>:MARKer<Mk>:REFerence:NAME....1049
CALCulate<Chn>:MARKer<Mk>:REFerence:SEARch:FORMat....1050
CALCulate<Chn>:MARKer<Mk>:REFerence:SEARch:LEFT....1628
CALCulate<Chn>:MARKer<Mk>:REFerence:SEARch:NEXT....1628

CALCulate<Chn>:MARKer<Mk>:REFerence:SEARch:RIGHt 1628
CALCulate<Chn>:MARKer<Mk>:REFerence:SEARch:TRACking 1052
CALCulate<Chn>:MARKer<Mk>:REFerence:SEARch[:IMMediate] 1628
CALCulate<Chn>:MARKer<Mk>:REFerence:TARGet 1054
CALCulate<Chn>:MARKer<Mk>:REFerence:THReshold 1055
CALCulate<Chn>:MARKer<Mk>:REFerence:THReshold:STATe 1055
CALCulate<Chn>:MARKer<Mk>:REFerence:TYPE 1056
CALCulate<Chn>:MARKer<Mk>:REFerence:X 1057
CALCulate<Chn>:MARKer<Mk>:REFerence:Y 1058
CALCulate<Chn>:MARKer<Mk>:REFerence[:STATe] 1053
CALCulate<Chn>:MARKer<Mk>:SEARch:FORMat 1050
CALCulate<Chn>:MARKer<Mk>:SEARch:LEFT 1628
CALCulate<Chn>:MARKer<Mk>:SEARch:NEXT 1628
CALCulate<Chn>:MARKer<Mk>:SEARch:RIGHt 1628
CALCulate<Chn>:MARKer<Mk>:SEARch:TRACking 1052
CALCulate<Chn>:MARKer<Mk>:SEARch[:IMMediate] 1628
CALCulate<Chn>:MARKer<Mk>:STYLe 1053
CALCulate<Chn>:MARKer<Mk>:TARGet 1054
CALCulate<Chn>:MARKer<Mk>:THReshold 1055
CALCulate<Chn>:MARKer<Mk>:THReshold:STATe 1055
CALCulate<Chn>:MARKer<Mk>:TYPE 1056
CALCulate<Chn>:MARKer<Mk>:X 1057
CALCulate<Chn>:MARKer<Mk>:Y 1058
CALCulate<Chn>:MARKer<Mk>[:STATe] 1053
CALCulate<Chn>:MATH:FORMatted:FUNCtion 1060
CALCulate<Chn>:MATH:FORMatted:STATe 1062
CALCulate<Chn>:MATH:FORMatted[:EXPRession]:SDEFine 1059
CALCulate<Chn>:MATH:FUNCtion 1060
CALCulate<Chn>:MATH:MEMorize 1061
CALCulate<Chn>:MATH:STATe 1062
CALCulate<Chn>:MATH:WUNit[:STATe] 1063
CALCulate<Chn>:MATH[:EXPRession]:SDEFine 1059
CALCulate<Chn>:MATH[:EXPRession][:DEFine] 1629
CALCulate<Chn>:PHOLd 1172
CALCulate<Chn>:PULSe:DUT:MODE 1075
CALCulate<Chn>:PULSe:EMPHasis:PRE 1076
CALCulate<Chn>:PULSe:EMPHasis:PST1 1076
CALCulate<Chn>:PULSe:EMPHasis:PST2 1076
CALCulate<Chn>:PULSe:EMPHasis:STATe 1077
CALCulate<Chn>:PULSe:EQUalization:CTLE:DC 1077
CALCulate<Chn>:PULSe:EQUalization:CTLE:POL1 1077
CALCulate<Chn>:PULSe:EQUalization:CTLE:POL2 1077
CALCulate<Chn>:PULSe:EQUalization:CTLE:RATio 1078
CALCulate<Chn>:PULSe:EQUalization:CTLE:ZERO 1078
CALCulate<Chn>:PULSe:EQUalization:STATe 1078
CALCulate<Chn>:PULSe:INPut:AMPLitude 1078
CALCulate<Chn>:PULSe:INPut:DRATe 1079
CALCulate<Chn>:PULSe:INPut:PWIDth 1079
CALCulate<Chn>:PULSe:INPut:SLICes 1079
CALCulate<Chn>:PULSe:MEASurement:CURSors:POST 1079

CALCulate<Chn>:PULSe:MEASurement:CURSors:PRE ... 1080
CALCulate<Chn>:PULSe:MEASurement:MAXTime ... 1080
CALCulate<Chn>:PULSe:MEASurement:STATe ... 1080
CALCulate<Chn>:PULSe:MEASurement:TYPE ... 1081
CALCulate<Chn>:PULSe:STATe ... 1081
CALCulate<Chn>:RIPPle:CLEar ... 1082
CALCulate<Chn>:RIPPle:CONTrol:DOMain ... 1082
CALCulate<Chn>:RIPPle:DATA ... 1083
CALCulate<Chn>:RIPPle:DELete:ALL ... 1084
CALCulate<Chn>:RIPPle:DISPlay[:STATe] ... 1084
CALCulate<Chn>:RIPPle:FAIL? ... 1085
CALCulate<Chn>:RIPPle:RDOMain:FORMat ... 1085
CALCulate<Chn>:RIPPle:SEGMent:COUNt? ... 1086
CALCulate<Chn>:RIPPle:SEGMent<Seg>:LIMit ... 1086
CALCulate<Chn>:RIPPle:SEGMent<Seg>:RESult? ... 1087
CALCulate<Chn>:RIPPle:SEGMent<Seg>:STIMulus:STARt ... 1088
CALCulate<Chn>:RIPPle:SEGMent<Seg>:STIMulus:STOP ... 1088
CALCulate<Chn>:RIPPle:SEGMent<Seg>[:STATe] ... 1088
CALCulate<Chn>:RIPPle:SOUNd[:STATe] ... 1089
CALCulate<Chn>:RIPPle:STATe ... 1089
CALCulate<Chn>:RIPPle:STATe:AREA ... 1089
CALCulate<Chn>:SMOothing:APERture ... 1173
CALCulate<Chn>:SMOothing[:STATe] ... 1174
CALCulate<Chn>:STATistics:DOMain:USER ... 1090
CALCulate<Chn>:STATistics:DOMain:USER:SHOW ... 1091
CALCulate<Chn>:STATistics:DOMain:USER:STARt ... 1091
CALCulate<Chn>:STATistics:DOMain:USER:STOP ... 1091
CALCulate<Chn>:STATistics:EPDelay[:STATe] ... 1092
CALCulate<Chn>:STATistics:FORMat ... 1092
CALCulate<Chn>:STATistics:MMPTpeak[:STATe] ... 1092
CALCulate<Chn>:STATistics:MSTDdev[:STATe] ... 1092
CALCulate<Chn>:STATistics:NLINear:COMP:LEVel ... 1092
CALCulate<Chn>:STATistics:NLINear:COMP:PHASe ... 1093
CALCulate<Chn>:STATistics:NLINear:COMP:RDOMain:USER ... 1093
CALCulate<Chn>:STATistics:NLINear:COMP:RDOMain:USER:SHOW ... 1093
CALCulate<Chn>:STATistics:NLINear:COMP:RDOMain:USER:STARt ... 1094
CALCulate<Chn>:STATistics:NLINear:COMP:RDOMain:USER:STOP ... 1094
CALCulate<Chn>:STATistics:NLINear:COMP:REFerence ... 1094
CALCulate<Chn>:STATistics:NLINear:COMP:RESult? ... 1094
CALCulate<Chn>:STATistics:NLINear:COMP:RLEVel ... 1095
CALCulate<Chn>:STATistics:NLINear:COMP:RMARker ... 1096
CALCulate<Chn>:STATistics:NLINear:COMP:RPHase ... 1096
CALCulate<Chn>:STATistics:NLINear:COMP[:STATe] ... 1096
CALCulate<Chn>:STATistics:RESult? ... 1097
CALCulate<Chn>:STATistics:RMS[:STATe] ... 1098
CALCulate<Chn>:STATistics:SFLatness[:STATe] ... 1098
CALCulate<Chn>:STATistics[:STATe] ... 1098
CALCulate<Chn>:STATistics[:STATe]:AREA ... 1099
CALCulate<Chn>:TRACking:MODE ... 1174
CALCulate<Chn>:TRACking:SWEep:POINt ... 1174

CALCulate<Chn>:TRANsform:COMPlex....1164
CALCulate<Chn>:TRANsform:DTFault:CENTer....1100
CALCulate<Chn>:TRANsform:DTFault:DEFine....1100
CALCulate<Chn>:TRANsform:DTFault:DELete....1101
CALCulate<Chn>:TRANsform:DTFault:PEAK:COUNt?....1102
CALCulate<Chn>:TRANsform:DTFault:PEAK:DATA<FaultNo>....1102
CALCulate<Chn>:TRANsform:DTFault:PEAK:STATe....1103
CALCulate<Chn>:TRANsform:DTFault:PEAK:THReshold....1103
CALCulate<Chn>:TRANsform:DTFault:POINts....1104
CALCulate<Chn>:TRANsform:DTFault:SELect....1104
CALCulate<Chn>:TRANsform:DTFault:SPAN....1100
CALCulate<Chn>:TRANsform:DTFault:STARt....1105
CALCulate<Chn>:TRANsform:DTFault:STATe....1105
CALCulate<Chn>:TRANsform:DTFault:STOP....1105
CALCulate<Chn>:TRANsform:IMPedance:RNORmal....1165
CALCulate<Chn>:TRANsform:PEEL:STATe....1165
CALCulate<Chn>:TRANsform:TIME:CAUSal....1106
CALCulate<Chn>:TRANsform:TIME:CENTer....1106
CALCulate<Chn>:TRANsform:TIME:DCHebyshev....1107
CALCulate<Chn>:TRANsform:TIME:EXPonential....1107
CALCulate<Chn>:TRANsform:TIME:GAUSsian....1107
CALCulate<Chn>:TRANsform:TIME:KAISer....1108
CALCulate<Chn>:TRANsform:TIME:LPASs....1108
CALCulate<Chn>:TRANsform:TIME:LPASs:DCSParam....1109
CALCulate<Chn>:TRANsform:TIME:LPASs:DCSParam:CONTinuous....1109
CALCulate<Chn>:TRANsform:TIME:LPASs:DCSParam:EXTRapolate....1110
CALCulate<Chn>:TRANsform:TIME:LPFRequency....1110
CALCulate<Chn>:TRANsform:TIME:RESolution:EFACtor....1110
CALCulate<Chn>:TRANsform:TIME:SPAN....1111
CALCulate<Chn>:TRANsform:TIME:STARt....1111
CALCulate<Chn>:TRANsform:TIME:STATe....1111
CALCulate<Chn>:TRANsform:TIME:STIMulus....1112
CALCulate<Chn>:TRANsform:TIME:STOP....1112
CALCulate<Chn>:TRANsform:TIME:WINDow....1113
CALCulate<Chn>:TRANsform:TIME:XAXis....1114
CALCulate<Chn>:TRANsform:TIME[:TYPE]....1113
CALCulate<Chn>:TTIMe:DATA....1166
CALCulate<Chn>:TTIMe:STATe....1166
CALCulate<Chn>:TTIMe:THReshold....1167
CONFigure:CHANnel:CATalog?....1175
CONFigure:CHANnel:MEASure:ACTive[:STATe]....1176
CONFigure:CHANnel:MEASure:ALL[:STATe]....1176
CONFigure:CHANnel:MEASure:OPTimized....1176
CONFigure:CHANnel<Ch>:MEASure[:STATe]....1177
CONFigure:CHANnel<Ch>:NAME....1178
CONFigure:CHANnel<Ch>:NAME:ID?....1178
CONFigure:CHANnel<Ch>:TRACe:CATalog?....1179
CONFigure:CHANnel<Ch>:TRACe:REName....1179
CONFigure:CHANnel<Ch>[:STATe]....1178
CONFigure:TRACe:CATalog?....1180

CONFigure:TRACe:MEM:ACTive ... 1181
CONFigure:TRACe:WINDow:TRACe? ... 1183
CONFigure:TRACe:WINDow? ... 1182
CONFigure:TRACe<Trc>:CHANnel:NAME:ID? ... 1181
CONFigure:TRACe<Trc>:CHANnel:NAME? ... 1180
CONFigure:TRACe<Trc>:NAME ... 1181
CONFigure:TRACe<Trc>:NAME:ID? ... 1182
CONFigure:TRACe<Trc>:REName ... 1182
CONTrol:AUXiliary:C[:DATA] ... 1185
CONTrol:GPIO:SENSe:SUMCurrent? ... 1187
CONTrol:GPIO<Port>:SENSe:CURRent? ... 1187
CONTrol:GPIO<Port>:SENSe:VOLTage? ... 1188
CONTrol:HANDler:A:MODE ... 1191
CONTrol:HANDler:A[:DATA] ... 1191
CONTrol:HANDler:B:MODE ... 1191
CONTrol:HANDler:B[:DATA] ... 1191
CONTrol:HANDler:C:MODE ... 1192
CONTrol:HANDler:C[:DATA] ... 1191
CONTrol:HANDler:D:MODE ... 1192
CONTrol:HANDler:D[:DATA] ... 1191
CONTrol:HANDler:E[:DATA] ... 1191
CONTrol:HANDler:F[:DATA] ... 1191
CONTrol:HANDler:G[:DATA] ... 1191
CONTrol:HANDler:H[:DATA] ... 1191
CONTrol:HANDler:INPut? ... 1193
CONTrol:HANDler:LOGic ... 1193
CONTrol:HANDler:OUTPut<Pt>:USER ... 1194
CONTrol:HANDler:OUTPut<Pt>[:DATA] ... 1193
CONTrol:HANDler:PASSfail:LOGic ... 1194
CONTrol:HANDler:PASSfail:MODE ... 1195
CONTrol:HANDler:PASSfail:POLicy ... 1196
CONTrol:HANDler:PASSfail:SCOPe ... 1196
CONTrol:HANDler:PASSfail:STATus? ... 1196
CONTrol:HANDler:RESet ... 1197
CONTrol:HANDler:SWEepend ... 1197
CONTrol:HANDler[:EXTension]:INDex:STATe ... 1192
CONTrol:HANDler[:EXTension]:RTRigger:STATe ... 1192
CONTrol:RFFE<Bus>:TEST:CLOCk:CURRent? ... 1200
CONTrol:RFFE<Bus>:TEST:CLOCk:VOLTage? ... 1202
CONTrol:RFFE<Bus>:TEST:DATA:CURRent? ... 1200
CONTrol:RFFE<Bus>:TEST:DATA:VOLTage? ... 1202
CONTrol:RFFE<Bus>:TEST:VIO:CURRent? ... 1200
CONTrol:RFFE<Bus>:TEST:VIO:VOLTage? ... 1202
CONTrol<Ch>:GPIO:SENSe:TRIGger ... 1187
CONTrol<Ch>:GPIO:TIME ... 1190
CONTrol<Ch>:GPIO:VOLTage:OUTPut ... 1190
CONTrol<Ch>:GPIO<Port>:RANGe ... 1186
CONTrol<Ch>:GPIO<Port>:SHUNt? ... 1188
CONTrol<Ch>:GPIO<Port>:VOLTage[:DEFault] ... 1190
CONTrol<Ch>:GPIO<Port>[:STATe] ... 1189

CONTrol<Ch>:RFFE:TEST:OUTPut....1202
CONTrol<Ch>:RFFE:TEST:SENSe:TRIGger....1202
CONTrol<Ch>:RFFE:TEST:TIME....1203
CONTrol<Ch>:RFFE<Bus>:COMMand:DATA....1197
CONTrol<Ch>:RFFE<Bus>:COMMand:SEND....1198
CONTrol<Ch>:RFFE<Bus>:COMMand:SEND?....1198
CONTrol<Ch>:RFFE<Bus>:SETTings:FREQuency....1198
CONTrol<Ch>:RFFE<Bus>:SETTings:VOLTage:HIGH....1199
CONTrol<Ch>:RFFE<Bus>:SETTings:VOLTage:IO....1199
CONTrol<Ch>:RFFE<Bus>:SETTings:VOLTage:LOW....1199
CONTrol<Ch>:RFFE<Bus>:SETTings[:STATe]....1199
CONTrol<Ch>:RFFE<Bus>:TEST:CLOCk....1200
CONTrol<Ch>:RFFE<Bus>:TEST:CLOCk:RANGe....1201
CONTrol<Ch>:RFFE<Bus>:TEST:CLOCk:SHUNt?....1201
CONTrol<Ch>:RFFE<Bus>:TEST:DATA....1200
CONTrol<Ch>:RFFE<Bus>:TEST:DATA:RANGe....1201
CONTrol<Ch>:RFFE<Bus>:TEST:DATA:SHUNt?....1201
CONTrol<Ch>:RFFE<Bus>:TEST:VIO....1200
CONTrol<Ch>:RFFE<Bus>:TEST:VIO:RANGe....1201
CONTrol<Ch>:RFFE<Bus>:TEST:VIO:SHUNt?....1201
CONTrol<Ch>:SEGMent<SegNr>:SEQuence:CLEar:ALL....1205
CONTrol<Ch>:SEGMent<SegNr>:SEQuence:COUNt?....1205
CONTrol<Ch>:SEGMent<SegNr>:SEQuence<SeqNr>:DELay....1203
CONTrol<Ch>:SEGMent<SegNr>:SEQuence<SeqNr>:GPIO<Port>:VOLTage....1204
CONTrol<Ch>:SEGMent<SegNr>:SEQuence<SeqNr>:RFFE<Bus>:COMMand:DATA....1204
CONTrol<Ch>:SEQuence:CLEar:ALL....1205
CONTrol<Ch>:SEQuence:COUNt?....1206
CONTrol<Ch>:SEQuence<SeqNr>:DELay....1206
CONTrol<Ch>:SEQuence<SeqNr>:GPIO<Port>:VOLTage....1206
CONTrol<Ch>:SEQuence<SeqNr>:RFFE<Bus>:COMMand:DATA....1207
DIAGnostic:DEVice:STATe....1208
DIAGnostic:DUMP:SIZE....1208
DIAGnostic:HUMS:BIOS?....1601
DIAGnostic:HUMS:DELete:ALL....1601
DIAGnostic:HUMS:DEVice:HISTory:DELete:ALL....1602
DIAGnostic:HUMS:DEVice:HISTory:EVENt:ADD....1603
DIAGnostic:HUMS:DEVice:HISTory?....1602
DIAGnostic:HUMS:EQUipment?....1603
DIAGnostic:HUMS:FORMat....1604
DIAGnostic:HUMS:SAVE....1604
DIAGnostic:HUMS:SECurity?....1604
DIAGnostic:HUMS:SERVice?....1605
DIAGnostic:HUMS:STATe....1606
DIAGnostic:HUMS:STORage?....1606
DIAGnostic:HUMS:SW?....1607
DIAGnostic:HUMS:SYSTem:INFO?....1608
DIAGnostic:HUMS:SYSTem:STATus:SUMMary?....1609
DIAGnostic:HUMS:SYSTem:STATus?....1609
DIAGnostic:HUMS:TAGS:ALL?....1610
DIAGnostic:HUMS:TAGS:DELete....1611

DIAGnostic:HUMS:TAGS:DELete:ALL ... 1611
DIAGnostic:HUMS:TAGS[:VALue] ... 1610
DIAGnostic:HUMS:UTILization:ACTivity:TRACking:STATe ... 1613
DIAGnostic:HUMS:UTILization:CUSTom:ADD ... 1613
DIAGnostic:HUMS:UTILization:CUSTom:ALL? ... 1614
DIAGnostic:HUMS:UTILization:CUSTom:DELete ... 1614
DIAGnostic:HUMS:UTILization:CUSTom:DELete:ALL ... 1615
DIAGnostic:HUMS:UTILization:CUSTom:UPDate ... 1615
DIAGnostic:HUMS:UTILization:HISTory:DELete:ALL ... 1617
DIAGnostic:HUMS:UTILization:HISTory:DETailed? ... 1617
DIAGnostic:HUMS:UTILization:HISTory? ... 1615
DIAGnostic:HUMS:UTILization? ... 1612
DIAGnostic:HUMS[:ALL]? ... 1600
DIAGnostic:PRODuct:OPTion:INFO? ... 1209
DIAGnostic:SERVice:CALibration:DATE ... 1619
DIAGnostic:SERVice:CALibration:DUE:DATE ... 1619
DIAGnostic:SERVice:CALibration:DUE:STATe? ... 1619
DIAGnostic:SERVice:CALibration:INTerval? ... 1619
DIAGnostic:SERVice:DATE ... 1620
DIAGnostic:SERVice:FUNCtion ... 1629
DIAGnostic:SERVice:HWINfo? ... 1209
DIAGnostic:SERVice:RFPower ... 1209
DIAGnostic:SERVice:SFUNction ... 1210
DIAGnostic:SERVice:STATe? ... 1620
DIAGnostic:SERVice:SWINfo? ... 1210
DISPlay:ANNotation:CHANnel[:STATe] ... 1212
DISPlay:ANNotation:INFO:POSition ... 1213
DISPlay:ANNotation:INFO:STATe ... 1213
DISPlay:ANNotation:MARKer:STIMulus[:STATe] ... 1213
DISPlay:ANNotation:MARKer:TRACe:INFO[:STATe] ... 1213
DISPlay:ANNotation:TRACe[:STATe] ... 1212
DISPlay:CMAP:BWSCheme[:STATe] ... 1214
DISPlay:CMAP:EYE:TCHigh:RGB ... 1215
DISPlay:CMAP:EYE:TCLow:RGB ... 1215
DISPlay:CMAP:LIMit:FCOLorize[:STATe] ... 1214
DISPlay:CMAP:LIMit:FSYMbol[:STATe] ... 1214
DISPlay:CMAP:LIMit[:STATe] ... 1214
DISPlay:CMAP:LSCHeme[:STATe] ... 1214
DISPlay:CMAP:LSSCheme[:STATe] ... 1214
DISPlay:CMAP:MARKer[:STATe] ... 1215
DISPlay:CMAP:TRACe:COLor[:STATe] ... 1217
DISPlay:CMAP:TRACe:RGB ... 1218
DISPlay:CMAP<DispEl>:RGB ... 1216
DISPlay:IWINdow:BFILter[:STATe] ... 1219
DISPlay:IWINdow:MARKer<Mk>[:STATe] ... 1219
DISPlay:IWINdow[:STATe] ... 1219
DISPlay:LAYout ... 1220
DISPlay:LAYout:APPLy ... 1220
DISPlay:LAYout:DEFine ... 1220
DISPlay:LAYout:EXECute ... 1221

DISPlay:LAYout:GRID....1222
DISPlay:LAYout:JOIN....1222
DISPlay:LAYout:OVERlay....1223
DISPlay:LAYout:SPLit....1223
DISPlay:MENU:KEY:ACTion:CATalog?....1223
DISPlay:MENU:KEY:EXECute....1223
DISPlay:MENU:KEY:SELect....1224
DISPlay:MENU:KEY:TOOL:CATalog?....1224
DISPlay:RFSize....1224
DISPlay[:WINDow<Wnd>]:ANNotation:MARKer:INFO[:STATe]....1224
DISPlay[:WINDow<Wnd>]:CATalog?....1225
DISPlay[:WINDow<Wnd>]:MARKer:CATalog?....1225
DISPlay[:WINDow<Wnd>]:MAXimize....1226
DISPlay[:WINDow<Wnd>]:NAME....1226
DISPlay[:WINDow<Wnd>]:OVERview[:STATe]....1226
DISPlay[:WINDow<Wnd>]:SIZE....1227
DISPlay[:WINDow<Wnd>]:TITLe:DATA....1227
DISPlay[:WINDow<Wnd>]:TITLe[:STATe]....1228
DISPlay[:WINDow<Wnd>]:TRACe:EFEed....1230
DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:CATalog?....1228
DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:COUPle....1228
DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:COUPle:ALL....1229
DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:DELete....1229
DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:FEED....1231
DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:LABel:SHOW....1231
DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:SHOW....1232
DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:X:OFFSet....1232
DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:Y:OFFSet....1233
DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:Y[:SCALe]:AUTO....1234
DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:Y[:SCALe]:BOTTom....1234
DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:Y[:SCALe]:CONTinuous....1235
DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:Y[:SCALe]:PDIVision....1236
DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:Y[:SCALe]:RLEVel....1236
DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:Y[:SCALe]:RPOSition....1237
DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:Y[:SCALe]:TOP....1234
DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:ZOOM:BOTTom....1238
DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:ZOOM:STARt....1239
DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:ZOOM:STOP....1239
DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:ZOOM:TOP....1238
DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:ZOOM[:STATe]....1239
DISPlay[:WINDow<Wnd>]:Y[:SCALe]:AUTO....1240
DISPlay[:WINDow<Wnd>][:STATe]....1227
FORMat:BORDer....1241
FORMat:DEXPort:SOURce....1630
FORMat[:DATA]....1241
HCOPy:DESTination....1242
HCOPy:DEVice:LANGuage....1243
HCOPy:ITEM:ALL....1244
HCOPy:ITEM:LOGO[:STATe]....1244
HCOPy:ITEM:MLISt[:STATe]....1244

HCOPy:ITEM:TIME[:STATe] 1245
HCOPy:PAGE:COLor 1245
HCOPy:PAGE:MARGin:BOTTom 1246
HCOPy:PAGE:MARGin:LEFT 1246
HCOPy:PAGE:MARGin:RIGHt 1246
HCOPy:PAGE:MARGin:TOP 1246
HCOPy:PAGE:ORIentation 1247
HCOPy:PAGE:WINDow 1247
HCOPy[:IMMediate] 1243
INITiate:CONTinuous:ALL 1248
INITiate:PIPelining:STATe 1250
INITiate[:IMMediate]:ALL 1249
INITiate<Ch>:CONTinuous 1247
INITiate<Ch>:STOP 1250
INITiate<Ch>[:IMMediate]:SCOPe 1250
INITiate<Ch>[:IMMediate][:DUMMy] 1249
INPut<PhyPt>:ATTenuation 1630
INSTrument:NSELect 1251
INSTrument:PORT:COUNt? 1251
INSTrument:SMATrix 1251
INSTrument:TPORt:COUNt? 1252
INSTrument[:SELect] 1631
MEMory:CATalog:COUNt? 1252
MEMory:CATalog? 1252
MEMory:DEFine 1253
MEMory:DELete:ALL 1253
MEMory:DELete[:NAME] 1253
MEMory:SELect 1253
MMEMory:AKAL:FACTory:CONVersion 1256
MMEMory:AKAL:USER:CONVersion 1256
MMEMory:CATalog:ALL? 1258
MMEMory:CATalog? 1257
MMEMory:CDIRectory 1258
MMEMory:CKIT:INFO? 1259
MMEMory:COPY 1259
MMEMory:DATA 1259
MMEMory:DELete 1260
MMEMory:DELete:CORRection 1260
MMEMory:FAVorite<FavId> 1260
MMEMory:LOAD:CABLe 1261
MMEMory:LOAD:CKIT 1261
MMEMory:LOAD:CKIT:SDATa 1262
MMEMory:LOAD:CKIT:SDATa:WLABel 1263
MMEMory:LOAD:CKIT:UDIRectory 1264
MMEMory:LOAD:CMAP 1265
MMEMory:LOAD:CMAP:HCOPy 1265
MMEMory:LOAD:CORRection 1266
MMEMory:LOAD:CORRection:MERGe 1266
MMEMory:LOAD:CORRection:RESolve 1267
MMEMory:LOAD:CORRection:TCOefficient<Ch> 1268

MMEMory:LOAD:EYE:BPATtern ... 1269
MMEMory:LOAD:EYE:EQUalization:CTLE ... 1269
MMEMory:LOAD:EYE:EQUalization:DFE ... 1270
MMEMory:LOAD:EYE:EQUalization:FFE ... 1270
MMEMory:LOAD:EYE:JITTer ... 1270
MMEMory:LOAD:EYE:MASK ... 1271
MMEMory:LOAD:LIMit ... 1271
MMEMory:LOAD:RIPPle ... 1272
MMEMory:LOAD:SEGMent ... 1273
MMEMory:LOAD:STATe ... 1274
MMEMory:LOAD:TRACe ... 1274
MMEMory:LOAD:TRACe:AUTO ... 1275
MMEMory:LOAD:VNETworks<Ch>:BALanced:DEEMbedding<LogPt> ... 1276
MMEMory:LOAD:VNETworks<Ch>:BALanced:EMBedding<LogPt> ... 1276
MMEMory:LOAD:VNETworks<Ch>:DIFFerential:EMBedding<LogPt> ... 1277
MMEMory:LOAD:VNETworks<Ch>:GLOop:DEEMbedding<group> ... 1278
MMEMory:LOAD:VNETworks<Ch>:GLOop:EMBedding<group> ... 1278
MMEMory:LOAD:VNETworks<Ch>:PPAir:DEEMbedding<ListId> ... 1279
MMEMory:LOAD:VNETworks<Ch>:PPAir:EMBedding<ListId> ... 1279
MMEMory:LOAD:VNETworks<Ch>:SENDed:DEEMbedding<PhyPt> ... 1280
MMEMory:LOAD:VNETworks<Ch>:SENDed:EMBedding<PhyPt> ... 1280
MMEMory:MDIRectory ... 1281
MMEMory:MOVE ... 1282
MMEMory:MSIS ... 1282
MMEMory:NAME ... 1282
MMEMory:RDIRectory ... 1283
MMEMory:SIMulation:DELete:ALL ... 1283
MMEMory:SIMulation:LOAD ... 1283
MMEMory:SIMulation:SDATa<Idx> ... 1284
MMEMory:SIMulation:SDATa<Idx>:DELete ... 1284
MMEMory:SIMulation:SDATa<Idx>:LOAD ... 1284
MMEMory:STORe:CABLe ... 1285
MMEMory:STORe:CKIT ... 1285
MMEMory:STORe:CKIT:WLABel ... 1286
MMEMory:STORe:CMAP ... 1286
MMEMory:STORe:CMAP:HCOPy ... 1286
MMEMory:STORe:CORRection ... 1287
MMEMory:STORe:CORRection:TCOefficient<Ch> ... 1287
MMEMory:STORe:EYE:EQUalization:CTLE ... 1288
MMEMory:STORe:EYE:EQUalization:DFE ... 1288
MMEMory:STORe:EYE:EQUalization:FFE ... 1289
MMEMory:STORe:EYE:MASK ... 1289
MMEMory:STORe:EYE:MASK:RESults ... 1289
MMEMory:STORe:EYE:MEASurements ... 1290
MMEMory:STORe:LIMit ... 1290
MMEMory:STORe:MARKer ... 1291
MMEMory:STORe:RIPPle ... 1291
MMEMory:STORe:SEGMent ... 1292
MMEMory:STORe:STATe ... 1292
MMEMory:STORe:TRACe ... 1292

MMEMory:STORe:TRACe:CHANnel....1294
MMEMory:STORe:TRACe:OPTion:BALanced....1295
MMEMory:STORe:TRACe:OPTion:COMMent....1295
MMEMory:STORe:TRACe:OPTion:DECimals:DATA....1296
MMEMory:STORe:TRACe:OPTion:DECimals:STIMulus....1296
MMEMory:STORe:TRACe:OPTion:EXTRapolate....1296
MMEMory:STORe:TRACe:OPTion:FORMat....1296
MMEMory:STORe:TRACe:OPTion:PLUS....1297
MMEMory:STORe:TRACe:OPTion:RSAMple....1297
MMEMory:STORe:TRACe:OPTion:RSAMple:POINts....1297
MMEMory:STORe:TRACe:OPTion:RSAMple:PRESet....1298
MMEMory:STORe:TRACe:OPTion:RSAMple:TIMStep....1298
MMEMory:STORe:TRACe:OPTion:SSEParator....1298
MMEMory:STORe:TRACe:OPTion:SYMMetric....1295
MMEMory:STORe:TRACe:OPTion:TABS....1298
MMEMory:STORe:TRACe:OPTion:TRIM....1299
MMEMory:STORe:TRACe:PORTs....1299
OUTPut:UPORt:ECBits....1301
OUTPut:UPORt:KEEP....1301
OUTPut<Ch>:UPORt:SEGMent<Seg>:STATe....1301
OUTPut<Ch>:UPORt:SEGMent<Seg>[:VALue]....1302
OUTPut<Ch>:UPORt[:VALue]....1303
OUTPut<Ch>[:STATe]....1300
OUTPut<Chn>:DPORt....1631
PROGram[:SELected]:EXECute....1305
PROGram[:SELected]:INIMessage....1306
PROGram[:SELected]:INIParameter....1306
PROGram[:SELected]:NAME....1307
PROGram[:SELected]:RETVal?....1308
PROGram[:SELected]:WAIT....1308
SOURce:POWer:CORRection:COLLect:AVERage:NTOLerance....1500
SOURce:POWer:CORRection:COLLect:AVERage[:COUNt]....1500
SOURce:POWer:CORRection:COLLect:CFACtor....1500
SOURce:POWer:CORRection:COLLect:FLATness....1501
SOURce:POWer:CORRection:COLLect:METHod....1501
SOURce:POWer:CORRection:COLLect:RRECeiver....1502
SOURce:POWer:CORRection:COLLect:VNETworks....1502
SOURce:POWer:CORRection:NREadings....1521
SOURce:POWer:CORRection:PMETer:ID....1521
SOURce:POWer:CORRection:PPOWer....1522
SOURce:POWer:CORRection:PSELect....1522
SOURce:POWer:CORRection:TCOefficient:COUNt?....1504
SOURce:POWer:CORRection:TCOefficient:DEFine<ListNo>....1505
SOURce:POWer:CORRection:TCOefficient:DELete:ALL....1505
SOURce:POWer:CORRection:TCOefficient:DELete<ListNo>[:DUMMy]....1506
SOURce:POWer:CORRection:TCOefficient:FEED....1507
SOURce:POWer:CORRection:TCOefficient:INSert<ListNo>....1507
SOURce:POWer:CORRection:TCOefficient[:STATe]....1508
SOURce:POWer:GENerator:SDELay....1525
SOURce:POWer:REDuce:SDELay....1528

SOURce:POWer:SWEepend:MODE 1530
SOURce:POWer:SWEepend:SDELay 1531
SOURce<Ch>:FREQuency:CONVersion:MIXer:PMFixed 1485
SOURce<Ch>:FREQuency:CONVersion:MIXer:PMODe 1486
SOURce<Ch>:FREQuency<PhyPt>:CONVersion:ARBitrary:EFRequency<Gen> 1483
SOURce<Ch>:FREQuency<PhyPt>:CONVersion:ARBitrary:IFRequency 1484
SOURce<Ch>:FREQuency<PhyPt>:FIXed 1487
SOURce<Ch>:FREQuency<PhyPt>[:CW] 1487
SOURce<Ch>:GROup:COUNt? 1489
SOURce<Ch>:GROup<Grp> 1488
SOURce<Ch>:GROup<Grp>:CLEar 1488
SOURce<Ch>:GROup<Grp>:DPORt:COUNt 1489
SOURce<Ch>:GROup<Grp>:NAME 1489
SOURce<Ch>:GROup<Grp>:PORTs 1490
SOURce<Ch>:GROup<Grp>:PPORt<PhyPort>:DPORt 1490
SOURce<Ch>:GROup<Grp>:PPORts 1491
SOURce<Ch>:GROup<Grp>:SIMultaneous:FOFFset:CONDition 1491
SOURce<Ch>:GROup<Grp>:SIMultaneous:FOFFset:MOFFset:BWFactor 1493
SOURce<Ch>:GROup<Grp>:SIMultaneous:FOFFset:MOFFset:DVALue 1493
SOURce<Ch>:GROup<Grp>:SIMultaneous:FOFFset:MOFFset:MODE 1494
SOURce<Ch>:GROup<Grp>:SIMultaneous:FOFFset:STATe 1495
SOURce<Ch>:LPORt<LogPt> 1531
SOURce<Ch>:LPORt<LogPt>:CLEar 1533
SOURce<Ch>:NFIGure:CALibration:APOWer 1497
SOURce<Ch>:NFIGure:CALibration:PPOWer 1497
SOURce<Ch>:NFIGure:CALibration:TPOWer 1497
SOURce<Ch>:NFIGure:PLEVeling:DGAin 1498
SOURce<Ch>:NFIGure:PLEVeling:DMAX 1498
SOURce<Ch>:NFIGure:PLEVeling:FATTenuation 1498
SOURce<Ch>:NFIGure:PLEVeling:LGAin 1498
SOURce<Ch>:NFIGure:PLEVeling:RATTenuation 1499
SOURce<Ch>:NFIGure:PLEVeling:RMAX 1499
SOURce<Ch>:NFIGure:PLEVeling:SATTenuation 1499
SOURce<Ch>:POWer:CORRection:DATA 1513
SOURce<Ch>:POWer:CORRection:DATA:PARameter<Wv>? 1513
SOURce<Ch>:POWer:CORRection:DATA:PORT<PhyPt> 1513
SOURce<Ch>:POWer:CORRection:GENerator<Gen>[:STATe] 1516
SOURce<Ch>:POWer:CORRection:IMODulation:PORT 1517
SOURce<Ch>:POWer:CORRection:NISTandards:DEFine 1519
SOURce<Ch>:POWer:CORRection:NISTandards:DELete 1520
SOURce<Ch>:POWer:CORRection:NISTandards:LOAD 1520
SOURce<Ch>:POWer:CORRection:OSOurces[:STATe] 1521
SOURce<Ch>:POWer:CORRection[:ACQuire] 1510
SOURce<Ch>:POWer:GENerator<Gen>:SLOPe 1525
SOURce<Ch>:POWer:GENerator<Gen>:STATe 1526
SOURce<Ch>:POWer<PhyPt>:CORRection:COLLect[:ACQuire] 1644
SOURce<Ch>:POWer<PhyPt>:CORRection:DATA:PARameter<Wv>:COUNt? 1515
SOURce<Ch>:POWer<PhyPt>:CORRection:GENerator<Gen>:LEVel:OFFSet 1515
SOURce<Ch>:POWer<PhyPt>:CORRection:IMODulation:LTONe[:ACQuire] 1517
SOURce<Ch>:POWer<PhyPt>:CORRection:IMODulation:UTONe[:ACQuire] 1518

SOURce<Ch>:POWer<PhyPt>:CORRection:LEVel:OFFSet ... 1518
SOURce<Ch>:POWer<PhyPt>:CORRection:STATe ... 1523
SOURce<Ch>:POWer<PhyPt>:CORRection:TCOefficient:CALibration ... 1503
SOURce<Ch>:POWer<PhyPt>:CORRection:TCOefficient:SWAP ... 1509
SOURce<Ch>:POWer<PhyPt>:CORRection[:ACQuire]:VERification:RESult? ... 1512
SOURce<Ch>:POWer<PhyPt>:GENerator<Gen>:OFFSet ... 1523
SOURce<Ch>:POWer<PhyPt>:GENerator<Gen>:PERManent[:STATe] ... 1524
SOURce<Ch>:POWer<PhyPt>:PERManent[:STATe] ... 1528
SOURce<Ch>:POWer<PhyPt>:REDuce[:STATe] ... 1529
SOURce<Ch>:POWer<PhyPt>:STARt ... 1530
SOURce<Ch>:POWer<PhyPt>:STATe ... 1529
SOURce<Ch>:POWer<PhyPt>:STOP ... 1530
SOURce<Ch>:POWer<PhyPt>[:LEVel][:IMMediate]:OFFSet ... 1527
SOURce<Ch>:POWer<PhyPt>[:LEVel][:IMMediate]:SLOPe ... 1527
SOURce<Ch>:POWer<PhyPt>[:LEVel][:IMMediate][:AMPLitude] ... 1526
STATus:PRESet ... 1534
STATus:QUEStionable:CONDition? ... 1534
STATus:QUEStionable:ENABle ... 1534
STATus:QUEStionable:INTegrity:CONDition? ... 1534
STATus:QUEStionable:INTegrity:ENABle ... 1534
STATus:QUEStionable:INTegrity:HARDware:CONDition? ... 1534
STATus:QUEStionable:INTegrity:HARDware:ENABle ... 1534
STATus:QUEStionable:INTegrity:HARDware:NTRansition ... 1535
STATus:QUEStionable:INTegrity:HARDware:PTRansition ... 1536
STATus:QUEStionable:INTegrity:HARDware[:EVENt]? ... 1535
STATus:QUEStionable:INTegrity:NTRansition ... 1535
STATus:QUEStionable:INTegrity:PTRansition ... 1536
STATus:QUEStionable:INTegrity[:EVENt]? ... 1535
STATus:QUEStionable:LIMit<Lev>:CONDition? ... 1534
STATus:QUEStionable:LIMit<Lev>:ENABle ... 1534
STATus:QUEStionable:LIMit<Lev>:NTRansition ... 1535
STATus:QUEStionable:LIMit<Lev>:PTRansition ... 1536
STATus:QUEStionable:LIMit<Lev>[:EVENt]? ... 1535
STATus:QUEStionable:NTRansition ... 1535
STATus:QUEStionable:PTRansition ... 1536
STATus:QUEStionable[:EVENt]? ... 1535
STATus:QUEue[:NEXT]? ... 1536
SYSTem:AGC:GLOBal:METHod ... 1565
SYSTem:COMMunicate:AKAL:CONNection ... 1538
SYSTem:COMMunicate:AKAL:MMEMory[:STATe] ... 1538
SYSTem:COMMunicate:CODec ... 1539
SYSTem:COMMunicate:GPIB[:SELF]:ADDRess ... 1539
SYSTem:COMMunicate:GPIB[:SELF]:DCLear:SUPPress ... 1539
SYSTem:COMMunicate:GPIB[:SELF]:INIT:WAIT ... 1539
SYSTem:COMMunicate:GPIB[:SELF]:LPORt:ALIGn ... 1540
SYSTem:COMMunicate:GPIB[:SELF]:RTERminator ... 1540
SYSTem:COMMunicate:NET:HOSTname ... 1541
SYSTem:COMMunicate:RDEVice:AKAL:ADDRess ... 1541
SYSTem:COMMunicate:RDEVice:AKAL:ADDRess:ALL? ... 1541
SYSTem:COMMunicate:RDEVice:AKAL:ADDRess:SUBModule:ALL? ... 1541

SYSTem:COMMunicate:RDEVice:AKAL:CKIT ... 1542
SYSTem:COMMunicate:RDEVice:AKAL:CKIT:CATalog? ... 1543
SYSTem:COMMunicate:RDEVice:AKAL:CKIT:STANdard:CATalog? ... 1544
SYSTem:COMMunicate:RDEVice:AKAL:DATE? ... 1544
SYSTem:COMMunicate:RDEVice:AKAL:FRANge? ... 1544
SYSTem:COMMunicate:RDEVice:AKAL:PORTs? ... 1545
SYSTem:COMMunicate:RDEVice:AKAL:PREDuction[:STATe] ... 1545
SYSTem:COMMunicate:RDEVice:AKAL:SCAN? ... 1546
SYSTem:COMMunicate:RDEVice:AKAL:SDATa? ... 1546
SYSTem:COMMunicate:RDEVice:AKAL:TEMPerature? ... 1547
SYSTem:COMMunicate:RDEVice:AKAL:WARMup[:STATe]? ... 1547
SYSTem:COMMunicate:RDEVice:GDEVice:LAN:STATe ... 1547
SYSTem:COMMunicate:RDEVice:GENerator:COUNt? ... 1548
SYSTem:COMMunicate:RDEVice:GENerator:DELete ... 1549
SYSTem:COMMunicate:RDEVice:GENerator:SCAN? ... 1550
SYSTem:COMMunicate:RDEVice:GENerator<Gen>:CATalog? ... 1547
SYSTem:COMMunicate:RDEVice:GENerator<Gen>:DEFine ... 1548
SYSTem:COMMunicate:RDEVice:GENerator<Gen>:SEPMode ... 1550
SYSTem:COMMunicate:RDEVice:GENerator<Gen>:SEPower ... 1550
SYSTem:COMMunicate:RDEVice:PMETer:CATalog? ... 1551
SYSTem:COMMunicate:RDEVice:PMETer:CONFigure:AUTO[:STATe] ... 1551
SYSTem:COMMunicate:RDEVice:PMETer:COUNt? ... 1552
SYSTem:COMMunicate:RDEVice:PMETer:DELete ... 1553
SYSTem:COMMunicate:RDEVice:PMETer:SCAN? ... 1553
SYSTem:COMMunicate:RDEVice:PMETer<Pmtr>:AZERo ... 1551
SYSTem:COMMunicate:RDEVice:PMETer<Pmtr>:DEFine ... 1552
SYSTem:COMMunicate:RDEVice:PMETer<Pmtr>:SPCorrection[:STATe] ... 1553
SYSTem:COMMunicate:RDEVice:SMATrix:CONFigure:ABORt ... 1554
SYSTem:COMMunicate:RDEVice:SMATrix:CONFigure:END ... 1554
SYSTem:COMMunicate:RDEVice:SMATrix:CONFigure:STARt ... 1555
SYSTem:COMMunicate:RDEVice:SMATrix:CONFigure:TVNA ... 1556
SYSTem:COMMunicate:RDEVice:SMATrix:COUNt? ... 1559
SYSTem:COMMunicate:RDEVice:SMATrix:DELete ... 1560
SYSTem:COMMunicate:RDEVice:SMATrix:SCAN? ... 1561
SYSTem:COMMunicate:RDEVice:SMATrix<Matr>:CATalog? ... 1554
SYSTem:COMMunicate:RDEVice:SMATrix<Matr>:CONFigure:MLTest ... 1557
SYSTem:COMMunicate:RDEVice:SMATrix<Matr>:CONFigure:MLVNa ... 1557
SYSTem:COMMunicate:RDEVice:SMATrix<Matr>:CONFigure:MTESt ... 1558
SYSTem:COMMunicate:RDEVice:SMATrix<Matr>:CONFigure:MVNA ... 1558
SYSTem:COMMunicate:RDEVice:SMATrix<Matr>:DEFine ... 1559
SYSTem:COMMunicate:RDEVice:SMATrix<Matr>:RELays:SWITch:COUNt? ... 1561
SYSTem:COMMunicate:RDEVice:SMATrix<Matr>:SERial? ... 1561
SYSTem:COMMunicate:REST:ENABle ... 1599
SYSTem:COMMunicate:SNMP:COMMunity:RO ... 1596
SYSTem:COMMunicate:SNMP:COMMunity:RW ... 1597
SYSTem:COMMunicate:SNMP:CONTact ... 1597
SYSTem:COMMunicate:SNMP:LOCation ... 1597
SYSTem:COMMunicate:SNMP:USM:USER ... 1598
SYSTem:COMMunicate:SNMP:USM:USER:ALL? ... 1598
SYSTem:COMMunicate:SNMP:USM:USER:DELete ... 1598

SYSTem:COMMunicate:SNMP:USM:USER:DELete:ALL ... 1599
SYSTem:COMMunicate:SNMP:VERSion ... 1599
SYSTem:CORRection:JOINcal ... 1565
SYSTem:CORRection:WIZard[:IMMediate] ... 1566
SYSTem:DATA:MEMory:ADD ... 1561
SYSTem:DATA:MEMory:CATalog? ... 1562
SYSTem:DATA:MEMory:COMMit ... 1562
SYSTem:DATA:MEMory:DELete ... 1563
SYSTem:DATA:MEMory:INITalize ... 1563
SYSTem:DATA:MEMory:OFFSet? ... 1563
SYSTem:DATA:MEMory:RESet ... 1563
SYSTem:DATA:MEMory:SIZE? ... 1564
SYSTem:DATE ... 1566
SYSTem:DFPRint? ... 1567
SYSTem:DISPlay:BAR:HKEY[:STATe] ... 1567
SYSTem:DISPlay:BAR:MENU[:STATe] ... 1567
SYSTem:DISPlay:BAR:STATus[:STATe] ... 1567
SYSTem:DISPlay:BAR:STOols[:STATe] ... 1567
SYSTem:DISPlay:BAR:TITLe[:STATe] ... 1567
SYSTem:DISPlay:BAR:TOOLs[:STATe] ... 1567
SYSTem:DISPlay:COLor ... 1567
SYSTem:DISPlay:CONDuctances ... 1568
SYSTem:DISPlay:DIALogs:SETup:MCAL[:STATe] ... 1568
SYSTem:DISPlay:TRACes:CCOunt ... 1568
SYSTem:DISPlay:UPDate ... 1568
SYSTem:DISPlay:XLABels ... 1569
SYSTem:ERRor:ALL? ... 1569
SYSTem:ERRor:COUNt? ... 1569
SYSTem:ERRor:DISPlay:ERRor ... 1570
SYSTem:ERRor:DISPlay:INFO ... 1570
SYSTem:ERRor:DISPlay:STATe ... 1570
SYSTem:ERRor:DISPlay:WARNings ... 1570
SYSTem:ERRor:DISPlay[:REMote] ... 1570
SYSTem:ERRor[:NEXT]? ... 1571
SYSTem:FIRMware:UPDate ... 1571
SYSTem:FORMat:IDENtify ... 1571
SYSTem:FPReset ... 1572
SYSTem:FREQuency? ... 1572
SYSTem:HELP:HEADers? ... 1573
SYSTem:HELP:SYNTax:ALL? ... 1573
SYSTem:HELP:SYNTax? ... 1573
SYSTem:IDENtify:FACTory ... 1573
SYSTem:IDENtify[:STRing] ... 1574
SYSTem:KLOCk ... 1574
SYSTem:LANGuage ... 1574
SYSTem:LOGGing:REMote[:STATe] ... 1575
SYSTem:OPCHeck:PLEVel:PORT<Port>? ... 1575
SYSTem:OPCHeck:SELFtest? ... 1576
SYSTem:OPTions:FACTory ... 1576
SYSTem:OPTions[:STRing] ... 1576

SYSTem:PASSword[:CENable] 1576
SYSTem:PRESet:REMote[:STATe] 1577
SYSTem:PRESet:SCOPe 1577
SYSTem:PRESet:STARt 1578
SYSTem:PRESet:USER:CAL 1578
SYSTem:PRESet:USER:NAME 1578
SYSTem:PRESet:USER[:STATe] 1579
SYSTem:PRESet[:DUMMy] 1577
SYSTem:REPort:ADD 1579
SYSTem:REPort:SAVE 1580
SYSTem:REPort:STARt 1580
SYSTem:SETTings:UPDate 1580
SYSTem:SHUTdown 1581
SYSTem:SMATrix:OPTimization 1582
SYSTem:SOUNd:ALARm[:STATe] 1582
SYSTem:SOUNd:STATus[:STATe] 1582
SYSTem:SRECorder:CLEar 1582
SYSTem:SRECorder:DATA? 1582
SYSTem:SRECorder[:STATe] 1582
SYSTem:TIME 1583
SYSTem:TRESet[:STATe] 1583
SYSTem:TSLock 1583
SYSTem:TTLout<Pt>:STATus[:STATe] 1584
SYSTem:USER:DISPlay:TITLe 1584
SYSTem:USER:KEY 1585
SYSTem:VERSion? 1585
TRACe:CLEar 1644
TRACe:COPY 1586
TRACe:COPY:MATH 1588
TRACe[:DATA]:STIMulus[:ALL]? 1645
TRACe[:DATA][:RESPonse][:ALL]? 1645
TRIGger:CHANnel<Ch>:AUXiliary:DURation 1594
TRIGger:CHANnel<Ch>:AUXiliary:INTerval 1595
TRIGger:CHANnel<Ch>:AUXiliary:OPOLarity 1595
TRIGger:CHANnel<Ch>:AUXiliary:POSition 1595
TRIGger:CHANnel<Ch>:AUXiliary[:ENABle] 1594
TRIGger[:SEQuence]:LTRigger:COUNt? 1590
TRIGger[:SEQuence]:LTRigger:RESet 1590
TRIGger[:SEQuence]:LTRigger[:STATe] 1590
TRIGger<Ch>[:SEQuence]:HOLDoff 1589
TRIGger<Ch>[:SEQuence]:LINK 1589
TRIGger<Ch>[:SEQuence]:MULTiple:HOLDoff 1591
TRIGger<Ch>[:SEQuence]:MULTiple:SLOPe<Num> 1591
TRIGger<Ch>[:SEQuence]:MULTiple:SOURce 1592
TRIGger<Ch>[:SEQuence]:SLOPe 1592
TRIGger<Ch>[:SEQuence]:SOURce 1593

Index

Symbols

[Center] 87
[Span] 87
[Start] 87
[Stop] 87
@DCL 1704
@GET 1704
@LOC 1704
@REM 1704
*.csv file format 151
*.limit 142
*.ripple 143
*.s<n>p file format 144
*CLS 904, 913
*ESE 913
*ESR? 913
*IST? 914
*OPC 914
*OPC (use) 901
*OPT? 914
*PCB 915
*PRE 915
*PSC 915
*RST 915
*SRE 915
*STB? 916
*TRG 916
*TST? 916
*WAI 916
2nd Mixer 328
µ1 21 (softkey) 317
µ2 21 (softkey) 317

A

a1 Source Port 1 306
a2 Source Port 2 306
AC supply 25
Accessories (connect) 29
Active Diagram 796
Active Power Cals dialog 657
Active Trace 450
Adapter Removal 194
Add Device
    external generators 853
    external power meters 848
    External switch matrices 858
Add Tr+Diag 451
Add Trace 451
Additional removable system drive
    Option R&S ZNB-B19 262
Addressed command 876, 1706
Adjust Time Gate 739
Admittance 125
Advanced (system settings) 818
    Error 820
    Info 820
    Show Messages 820
    Warning 820
AGC Manual Configuration dialog 701
    All Segments – Learn Sweep 702
    All Segments – Reset 702
    Drive-port specific settings 702
    Range 702
    Set All Items to Auto 703
    Set All Items to Low Distortion 703
    Set All Items to Low Noise 703
AGC Mode
    Low Dist 701
    Low Noise 701
Align *RST to Use Defined Preset 813
All Channels Continuous 560
All Channels on Hold 560
All Data & Func to <Destination> 458
All Data to <Destination> 458
All Partial Meas'ments 538
All S-Params 294
Annexes 1700
Aperture 134
Aperture Points 443
Applic softtool 794
    External Tools application 794
    Eye Diag tab 381
    Eye Mask Test tab 397
    Rise Time tab 402
    TDR Setup tab 375
    Time Gate tab 405
Application cards 20
Application notes 20
Apply
    SMARTerCal 626
    System error correction 591
Apply (calibration) 598
Arbitrary port settings
    R&S ZNB-K4 233
ASCII 810
ASCII (*.csv) file 151
ATN 1705
Attention 1705
Auto Config 849
Auto Length 740
Auto Length and Loss 744
Auto Power Reduction for Cal Unit 813
Auto scale 67
Auto Scale Diagram 445
Auto Scale Trace 445
Auto Zero 319
Automatic calibration 175
Automatic Harmonic Grid 363
Average 535
Average tab 534
Averaging mode 535
    Auto 535
    Flatten Noise 535
    Moving Average 535
    Reduce Noise 535
Axis Pair 364

B

b1 / a1 Source Port 1 303
b1 / a2 Source Port 2 303
b1 Source Port 1 306
b1 Source Port 2 306
b2 / a1 Source Port 1 303

b2 / a2 Source Port 2 303
b2 Source Port 1 306
b2 Source Port 2 306
Balanced Params 479
Balanced port 130
Balanced Ports dialog 296
Logical Port Config tab 298
Predefined Config tab 297
Reference Impedance tab 300
User Config tab 299
Band pass impulse 219
Bandfilter search 102, 518
Bandpass Ref to Max 519
Bandpass Ref to Mkr 520
Bandstop Ref to Max 520
Bandwidth 519
Result Off 521
Search bandpass or bandstop 521
Search Mode 521
Tracking 522
Bandpass 102
Bandpass (time gate) 365
Bandstop 102
Bandwidth 534
Bandwidth (bandfilter) 102
Bandwidth tab 533
IF Filter (digital) 534
Basic instrument concepts 79
Basic tasks 44
Bias tees 44
Option R&S ZNB-B1 258
Black White Scheme 806
Block data format 880
Boolean parameter 880
Brochure 20
Buffered Sweep 1670

C

Cal Connector Types dialog 634
Char. Imp. 635
Cutoff Freq. fc 636
Line Type 636
Offset model 636
Rel. Permittivity εr 636
Cal group 174
Cal Kit
Import 587, 623
Selection 586, 622
Cal kit file 173
Cal Kits
Calibration Setting wizard 584
SMARTerCal Wizard (Manual, MultiCal) 630
Cal Power Config dialog 650, 691
Cal Power Offset 652
Cal softtool 562
Cal Devices tab 633
METAS Cal tab 660
Power Cal Settings tab 648
Start Cal tab 562
Use Cal tab 655
Cal Unit
Calibration 568, 579, 604, 615
Characterization 568, 579, 605, 616
Cal Validation info 831
Calibrate all Channels 584
Calibration 154, 562, 1673
Cal Devices 633
Cal Unit 568, 579, 604, 615
Cal Unit Characterization 568, 579, 605, 616
Calibrate all Channels 584
Devices 633
Intermodulation 342
Manual 581
Ports 567, 579, 583, 594, 663
Power calibration settings 648
Procedure 562
Program example 1673
SMARTerCal (manual) 617
Source port 584, 594
Sweep diagrams 575, 588, 624
Type 569, 580, 584, 594
Type (METAS) 664
Validation 183
Calibration (De-)Embedding dialog 653
Two Port Configuration dialog 654
Calibration (manual)
Connector 586, 622
Same Connector All Ports 587, 623
Same Gender All Ports 587, 623
Sweep 589
Calibration (system settings) 813
Calibration kit 168
Calibration Kits dialog 637
Add cal kit 638
Available Cal Kits 638
Connector Type 638
Copy cal kit 638
Delete cal kit 638
Export Cal Kit 639
Import Cal Kit 639
Standards 638
Calibration labels
Scalar power calibration 191
SMARTerCal 195
System error correction 175
Calibration Manager dialog 657, 658
Add 658
Apply 658
Apply to All 658
Channel and calibration properties 659
Channel State 658
Delete from Pool 659
Pool 659
Preset User Cal 659
Replace 658
Resolve Pool Link 659
Calibration pool 174
Calibration Presetting (dialog)
SMARTerCal 627
Calibration Setting wizard 581
Apply 591
Cal Kit 586, 622
Calibration 587
Connectors and Cal Kits 584
Ports and Type 582
Calibration Setting wizard (MultiCal) 591
Calibration 595
Connectors and Cal Kits 594
Define Calibration 592
Ports and Type 592
Calibration standard 168

Calibration Type
SMARTerCal ... 620, 629
SMARTerCal (Auto) ... 605, 616
Calibration types ... 155
Calibration unit ... 175, 183
Calibration Unit
Detect Assignment ... 572, 575, 607, 611
Detect Ports & Start Cal ... 575, 610
Set to Default Port Assignment ... 572, 608
Calibration Unit (dialog)
SMARTerCal ... 330, 332
Calibration Unit wizard ... 565
Calibration Type ... 569, 580
Calibration Unit wizard (MultiCal) ... 576
Define Calibration ... 578
Cartesian diagram ... 108
Center (bandfilter) ... 102
Center = Marker ... 523
Center Frequency ... 527
Channel ... 80
programming examples ... 1650
Channel Bits (system settings) ... 817
Decimal ... 817
Pin 16 - 19 ... 818
Channel Config
Define Parallel Measurement ... 694
Mixer Mode ... 323
Port Config ... 685
Port Settings ... 686
Channel Config softtool ... 670
Channels tab ... 670
Intermodulation tab ... 334
Mixer Mode tab ... 323
Mode tab ... 698
Port Config tab ... 685
Pwr Cal Settings tab ... 703
Channel Info ... 802
Channel keys
[Cal] ... 37
[Channel Config] ... 37
[Offset Embed] ... 37
[Pwr Bw Avg] ... 37
[Sweep] ... 37
[Trigger] ... 37
Channel Manager dialog ... 673
Add channel ... 674
Channel table ... 673
Delete channel ... 674
Channel properties ... 659
Channel settings ... 103
Channels tab
Active Channel ... 671
Channel Manager ... 673
Channel On ... 672
Controls ... 670
Copy Channel ... 671
Copy Channel + Diagram ... 672
Delete Channel ... 672
Fixture Simulator ... 672
New Channel ... 671
New Channel + Diagram ... 671
Characterization
Cal Unit ... 568, 579, 605, 616
Characterization Data ... 645
Characterization wizard ... 645
Test Port Assignment ... 646
Characterize Cal Unit dialog ... 643
Authentication ... 644
Set Password ... 644
Chirp z-transformation ... 219
Circle limit ... 139
Circle Test ... 497
Clear Test ... 500
Define circle ... 500
Draw Circle ... 500
Limit Check ... 498
Show Limit Circle ... 498
Circular diagrams ... 68
Clear All
SCPI recorder ... 907
Clear status
Remote ... 913
CMRR measurement ... 314
CMRR12 (softkey) ... 315
CMRR21 (softkey) ... 315
Color Scheme ... 801
Colorize Trace when Failed ... 805
Command
Common ... 876
Device-specific ... 876
Processing ... 886
Recognition ... 886
Sequence ... 886
Structure and syntax ... 876
Synchronization ... 886
Command reference ... 917
Common commands ... 911, 913
Common mode ... 129, 131
Common Mode ... 302
Compatibility mode
Sweep Active Channel ... 562
Sweep All Channels ... 562
Computer name
Changing ... 33
CONDition ... 892, 893
Configured devices
external generators ... 852
external power meters ... 846
External switch matrices ... 856
Connectors ... 1701
Calibration Setting wizard ... 584
SMARTerCal Wizard (Manual, MultiCal) ... 630
Context menu ... 104
Continuous sweep ... 559
Control bus ... 1705
Control elements ... 48
Control lines (GPIB bus) ... 1704
Conversion (mixer) ... 327
Couple Active Trace ... 447
Couple All Traces ... 447
Customer support ... 1692
CW Frequency ... 528
CW Mode Spectrum ... 335
CW Mode sweep ... 542

D

Data
Accessing ... 1693
Handling ... 1693
Management ... 1693
Retrieving ... 1693
Data & Func to <Destination> ... 457

Data base ... 886
Data bus (GPIB) ... 1704
Data entry ... 38, 61
Data flow ... 89
Data sheet ... 20
Data to <Destination> ... 456
Data Valid ... 1705
DAV ... 1705
dB Mag ... 438
DC Iinputs ... 44
DC INPUT 1 ... 4 ... 319
DC inputs
Option R&S ZNB-B81 ... 264, 319
DC Point Extrapolation ... 479
DC softtool tab ... 319
DC Value ... 362
DC1 (softkey) ... 320
DC2 (softkey) ... 320
DC3 (softkey) ... 320
DC4 (softkey) ... 320
DCL ... 1704, 1706
DCL, SDC ... 904
Decouple Active Trace ... 447
Decouple All Traces ... 447
Deembedding
Balanced ... 710, 711
Ground Loop ... 711, 712
Impedance Renormalization ... 712, 713
Port Sets ... 708, 710
Single Ended ... 707, 708
DEFault ... 880
Default file locations ... 1254
Default values
Remote ... 915
Define Calibration dialog
Calibration Setting wizard (MultiCal) ... 578, 592
Define Limit Lines dialog ... 488
Define Mixer Meas dialog ... 326
Define Parallel Measurement dialog ... 694
Balanced Ports tab ... 696
DUT Definition tab ... 695
Frequency Offset tab ... 696
Minimum Offset ... 697
Modify an existing DUT ... 695
Parallel Measurement with Frequency Offset ... 697
Reference Impedance tab ... 696
State indication popup ... 698
Define Physical Ports dialog ... 864
Define Ripple Test dialog ... 495
Define Segments dialog ... 545
Displayed Columns ... 549
Define User Color Scheme dialog ... 803
Delay ... 134
Delay (Offset Embed) ... 738, 739
Delete All Cal Kits ... 814
Delete All Mem ... 459
Delete Cal Pool ... 814
Delete Trace ... 451
Delta Mode ... 505
Delta-L 4.0 PCB characterization
R&S ZNB-K231 ... 256
Denominator ... 305
Destination (memory trace) ... 457
Destination (memory traces) ... 458
Detector ... 127
Device address ... 874
Device Control
Option R&S ZNB-B12 ... 259
Device LAN Configuration dialog ... 858
Device message ... 876
Device response ... 876
Dflt Marker Frmt ... 443
DHCP server
LAN configuration ... 33
Diagram ... 57, 80, 795
Active Diagram ... 796
Add Tr+Diag ... 796
Delete Diagram ... 797
Maximize Diagram ... 797
Name ... 797
Overlay All ... 798
programming examples ... 1650
Show Title ... 797
Title ... 797
Trace List ... 796
Dialog ... 104
Dialogs ... 56
Differential imbalance measurement ... 316
Differential mode ... 129, 131
Differential Mode ... 302
Direct control interface ... 44
Display ... 36
Diagram ... 795
Display Circle ... 501
Display Config ... 801
Channel Info ... 802
Color Scheme ... 801
Font Size ... 803
Hide Sensitive Information ... 802
Info Table ... 803
Trace Info ... 802
Display elements ... 93
Display softtool ... 795
Config tab ... 801
Diagram tab ... 795
Split tab ... 799
Touchscreen tab (R&S ZNB only) ... 809
View Bar tab ... 808
Displayed Columns dialog ... 549
Distance Center ... 530
Distance Span ... 530
Distance Start ... 530
Distance Stop ... 530
Distance to Fault
Sweep points ... 368
Window ... 367
Distance to Fault tab ... 366
Distance-to-fault measurements
Option R&S ZNB-K3 ... 231, 366
DNS server
LAN configuration ... 33
Dolph-Chebyshev
Side Lobe Level ... 359
DOWN ... 880
Drag and drop ... 60
Driver (remote control) ... 870
Driving Mode
Alternated ... 699
Auto ... 699
Chopped ... 699

E

Eazy de-embedding (IEEE 370)
R&S ZNB-K210 ... 253
Electrical Length ... 134, 738, 739
Electrostatic discharge (ESD) ... 43
Embedding
Balanced ... 710, 711
Differential Match ... 713, 714
Ground Loop ... 711, 712
Impedance Renormalization ... 712, 713
Port Sets ... 708, 710
Single Ended ... 707, 708
EMI suppression ... 24
ENABle ... 890, 892
Enable registers
Remote ... 915
EOI ... 1705
Error log ... 830
Error message ... 1687
Error queue ... 890, 903
Errors ... 1687
Asynchronous ... 1688
During firmware update ... 1689
During measurement ... 1688
FPGA update ... 1689
ESB ... 895
ESE ... 890, 896
ESR ... 896
Evaluation of data ... 75
Evaluation Range Dialog ... 470
EVENt register ... 892
Event status enable register (ESE)
Remote ... 913
Event status register (ESR)
Remote ... 913
Execution (command)
Overlapping ... 886
Sequential ... 886
Export
Sweep segments ... 548
Export As
SCPI recorder ... 907
Export Data - <File Type> dialog ... 477
Ask Before Overwriting ... 479
Comment Added to File ... 482
Contents ... 479
Output Format ... 479
Export Data - ASCII Files dialog
Add Ref Impedances ... 480
Decimal Separator ... 481
Field Separator ... 481
Formatted Values ... 480
Resample ... 481
Export Data - MatLab Files dialog
Add Ref Impedances ... 480
Field Separator ... 481
Formatted Values ... 480
Resample ... 481
Export Data - Touchstone Files dialog
Balanced Params ... 479
DC Point Extrapolation ... 479
Normalize with Port-Specific Impedances ... 479
Export Markers ... 508
Export snp Files
Active Channel ... 483
All Channels ... 483
Assign port numbers ... 483
Reset Port Assignments ... 483
Select ports ... 483
Select Ports ... 482
Extended dynamic range
Options R&S ZNB4-B52, R&S ZNB4-B54 ... 263
Options R&S ZNB8-B52, R&S ZNB8-B54 ... 263
Extended power range
Options R&S ZNBxx-B22, R&S ZNBxx-B24 ... 262
Extended time domain analysis
Option R&S ZNB-K20 ... 226
External Frequency ... 842
External generator ... 269
External Generators dialog ... 851
External Matrices dialog ... 855
External power meter ... 267
External Power Meter Config dialog ... 849
External Power Meters dialog ... 845
External reference ... 842
Eye Diag ... 381
Eye diagram
Basic settings ... 383
Display measurements ... 381
DUT ... 393
Emphasis ... 388
Enable/disable ... 381
Equalization ... 394
Export measurements ... 383
Generator bit pattern ... 388
Generator bit stream ... 384
Generator bit stream file ... 385
Generator bit stream length ... 384
Generator encoder ... 388
Generator levels ... 386, 388
Generator low pass ... 386
Generator modulation ... 385, 388
Generator rise time ... 386, 388
Generator scrambler ... 388
Generator settings ... 387
Generator symbol rate ... 385
Jitter ... 390
Mask configuration ... 398
Mask test ... 397
Noise ... 392
Select displayed measurements ... 382
Eye Diagram simulation
Advanced settings ... 387
Eye Mask Test
Define Mask Configuration ... 398
EZD
Advanced settings ... 725

F

Factor (averaging) ... 535
Factory calibration ... 1700
Favorites ... 783
File
Management ... 1693
File softtool ... 778
Favorites tab ... 783
More tab ... 787
Print tab ... 784, 792
Recall Sets tab ... 779
Trace Data tab ... 787
Firewall ... 35

Firmware
Installation ... 1701
Setup file ... 1701
Firmware installation
Admin account ... 1700
Firmware update ... 1700
First Partial Meas'ment ... 538
Fixture Compensation
Auto Length ... 741
Auto Length and Loss ... 741
Direct Compensation ... 742
Fixture Compensation dialog ... 741
Flatness Cal
Convergence ... 649
Max Iterations ... 649
Tolerance ... 649
Flatness calibration ... 188
Font Size ... 803
Format softtool ... 438
Formula (PAE) ... 322
FPGA update failure ... 1689
Freq for Loss ... 706, 744
Freq Step Size ... 537
Freq Sweep Mode
Stepped ... 539
Swept ... 539
Frequency ... 342
Frequency (mixer) ... 328
Frequency conversion measurements
R&S ZNB-K4 ... 233
Frequency reference ... 841
Frequency resolution 1 mHz
Option R&S ZNB-K19 ... 244
Front panel ... 36
Front panel keys ... 45
Fullscreen mode ... 808
Function keys ... 37
Channel section ... 37
Stimulus section ... 38
System section ... 38
Trace section ... 37
Fuse replacement ... 1698

G

Generator (external) ... 269
GET ... 1704, 1706
Getting started ... 19
Global settings ... 79
GPIB
Introduction ... 868
GPIB Address ... 843
GPIB bus control
Remote ... 915
GPIB bus interface ... 1704
GPIB bus messages ... 876
GPIB Explorer ... 794, 870
GPIB interface ... 44
GPIB Interface
R&S ZNB-B10 ... 259
GPIO configuration
Apply (Set) ... 681
Basic ... 680
Output Voltage ... 681
Seq. ... 681
Voltage ... 681
Voltage and current measurements ... 681
GPIO interface
external ... 44
Option R&S ZN-B15 ... 260, 674, 1715
GPIO voltage and current measurements ... 681
Range ... 682
Results ... 683
Shunt ... 682
Start Meas ... 682
Graphic zoom ... 65
Ground connector ... 43
Group Delay ... 442
Aperture Points ... 443
GTL ... 1706

H

Handler I/O ... 1707
Option R&S ZN-B14 ... 260, 1707
Handler I/O (universal interface) ... 44
Handshake bus ... 1704
Hardkey panel ... 50
Hardware ... 829
Hardware (connectors) ... 1701
Hardware info ... 829
Harmonic grid ... 221
Automatic ... 363
Set and keep ... 361
Health and utilization monitoring service
HUMS ... 257
Help
SCPI recorder ... 907
Hide All Data ... 459
Hide All Mem ... 459
Hide All Other Traces ... 452
Hide Sensitive Information ... 802
HiSLIP ... 868
Horiz. Line ... 502
Response Value ... 502
Show Horiz. Line ... 502
HUMS
R&S ZNB-K231 ... 257

I

Ideal (cal kit) ... 171
Identification
Remote ... 914
IEC/IEEE Bus Interface ... 1704
IECWIN32 ... 870
IEEE 488 CH 1 ... 1704
IEEE 488 CH 2 ... 1704
IF Filter (digital) ... 534
IFC ... 1705, 1706
Imag ... 442
Image Suppr.
Auto ... 699
LO < RF ... 699
LO > RF ... 699
Imb12 (softkey) ... 315
Imb21 (softkey) ... 315
Imbal. CMRR tab ... 314
Imbalance (softkey) ... 315
Imbalance Differential dialog ... 316
Imbalance measurement ... 314
Imbalance parameters ... 132
Impedance ... 121

Import
Cal Kit ... 587, 623
Sweep segments ... 548
Import Complex Data dialog ... 476
Import Data to New Mem ... 477
Import Data to New Mem ... 477
Impulse Response ... 359
In-situ de-embedding
R&S ZNB-K220 ... 255
INFintiy ... 880
Info dialog ... 826
Cal Validation tab ... 831
Hardware tab ... 829
Lost Trigger tab ... 831
Options tab ... 828
Selftest tab ... 830
Setup tab ... 828
Info Table ... 803
Inline calibration ... 179, 1680
Instrument calibration ... 73
Instrument hardware ... 886
Instrument Messages ... 844, 1707
Instrument security procedures ... 20
Instrument tour ... 35
Instrument-control commands ... 876
Interface
Functions (GPIB bus) ... 1706
Interface Clear ... 1705
Interface message ... 876
Interface messages
GPIB ... 1706
VXI-11 ... 1704
Interfaces ... 1701
Intermod tab
IM Products dialog ... 346
Intercept Points dialog ... 347
Intermodulation
Define ... 338
Intermodulation measurements ... 344
Higher order ... 346
Intercept points ... 347
Measurements tab ... 344
Option R&S ZNB-K14 ... 238
Results ... 240
Internal reference ... 842
Inv Smith ... 442
Inverted Smith chart ... 113
IP Address ... 842
IP address setting ... 31
ISD
Advanced settings ... 720
IST flag ... 890, 896
Remote ... 914

K

K 21 (softkey) ... 317
Keyboard ... 30
Keyword (SCPI command) ... 876
Kit Standards dialog ... 639
Add standard ... 641
Copy standard ... 641
Delete standard ... 641
One Port Standards ... 640
Read .s1p File ... 641
Read .s2p File ... 641
Two Port Standards ... 640
View / Modify standard ... 641
Known devices
external generators ... 852
external power meters ... 846
External switch matrices ... 856

L

LAN ... 43
Remote operation ... 31
LAN Detection
external generators ... 853
external power meters ... 847
External switch matrices ... 857
Language ... 810
Light Scheme ... 806
Limit line ... 135
files ... 142
Limit Test ... 484, 488
Add / Insert / Delete segment ... 490
Clear Test ... 486
Delete All segments ... 490
Limit Check ... 485
Limit line from trace data ... 490
Recall limit line ... 490
Save limit line ... 490
Segments ... 489
Shift Lines ... 488
Show Limit Line ... 485
Limit Test tab
Controls ... 484
Limits
Global Check ... 486
TTL1 Pass ... 487
TTL2 Pass ... 487
Lin Freq ... 540
Lin Mag ... 441
Line Styles Scheme ... 806
Lines softtool ... 484
Circle Test tab ... 497
Display Circle tab ... 501
Horiz. Line tab ... 502
Limit Test tab ... 484
Ripple Test tab ... 493
List of recorded commands/script editor
SCPI recorder ... 906
LLO ... 1706
Load Match Correction ... 656
Load Recording
SCPI recorder ... 906
LOC ... 1704
Log Errors ... 845, 851
Log Freq ... 540
Log Mag ... 441
Logical Port
Offset Embed Differential Match ... 758
Loss
Active ... 706
Loss (bandfilter) ... 102
Loss at DC ... 706, 744
Loss at Freq ... 706, 744
Loss parameters
definition ... 198
Lost trigger
Tab ... 831
Lost Trigger info ... 831

Low pass impulse 219
Low Pass Settings dialog 361
Low pass step 219
Lower edge (bandfilter) 102
Lower limit 135
Lower Tone (intermodulation) 339

M

Maintenance 1698
Marker 57, 96
  Delta Mode 505
  Discrete 507
  Format 507
  Info 508
  Mkr 1 ... Mkr 10 505
  Mode 507
  Name 506
  On 504
  Properties 506
  Props 506
  Ref Mkr -> Mkr 508
  Response 504
  Search 503
  Set by 68
  Settings 503
  Stimulus 504
  Style 507
Marker coupling 101
Marker Format 507
Marker Info 508
Marker info field 98
Marker Mode
  Arbitrary 507
  Fixed 507
  Normal 507
Marker Name 506
Marker Search 509
  Max / Min 509
  Next Peak 510
  Peak Left / Peak Right 510
  Peak Type 510
  Search Mode 518
Marker search functions 101
Marker softtool 503
  Bandfilter tab 518
  Info Field tab 524
  Marker Coupling tab 525
  Marker Props tab 506
  Marker Search tab 509
  Markers tab 503
  Multiple Peak tab 514
  Set by Marker tab 522
  Target Search tab 516
Marker Style 507
Markers
  All Off 504
  Coupled Markers 506, 525
  Decimal Places 508
  Export 508
  Ref Mkr 505
Max (softkey, trace scale) 446
Max = Marker 523
MAXimum 880
Meas
  Power Sensor 318
  S-Params 292
Meas Delay 538
Meas softtool 291
  DC tab 319
  Imbal. CMRR tab 314
  Power Sensor tab 318
  Ratios tab 302
  S-Params tab 292
  Skew tab 403
  Stability tab 317
  TDR measurements 374
  Wave tab 305
  Y-Z-Params tab 312
  Y←Sij tab 311
  Z←Sij tab 308
Measure Fixture 742
Measurement examples 70
Measurement receiver calibration 189
Measurement results 116
Measurement Type (PAE) 321
Measurement uncertainty analysis
  Option R&S ZNB-K50/R&S ZNB-K50P 248, 660
Measurements 291
  CMRR 314
  Converted admittances 311
  Converted impedances 308
  DC values 319
  Differential imbalance 316
  Imbalance 314
  Intermodulation parameters 344
  Intermodulation products 346, 347
  Port activation on demand 292
  Power (with external power meter) 318
  Ratios of wave quantities 302
  S-parameters 292
  Stability factors K, µ1, µ2 317
  Wave quantities 305
  Y-parameters 312
  Z-parameters 312
Mechanical Length 738, 739
Mem Math tab
  Controls 460
Menu bar 52
Menu Bar
  Show/hide 808
METAS Cal tab 660
METAS Calibration
  Type 664
Min (softkey, trace scale) 446
Min = Marker 523
Minimize application 28
MINimum 880
Mixed mode parameters 131
Mixer Mode 323
Mixer Mode tab
  Controls 323
Mode conversion factor 131
Mode tab
  AGC Manual Configuration dialog 701
  AGC Mode 701
  Driving Mode 699
  Image Suppr. 699
  Segmented AGC 701
Modify Frequency Conversion dialog 688
  Frequency conversion formula 689
Monitor 29
Monitor (DisplayPort) 43
Monitor (DVI-D) 43

More Ratios dialog 304
More Wave Quantities dialog 307
Mouse 30
Multiple Peak search 514
Eval Range 515
Max 514
Min 514
Tracking 515
Multiple triggers
Source 556
Multiplication (mixer) 329

N

NAN (not a number) 880
Navigation keys 39
Navigation tools 48
NDAC 1705
New (recall set) 780
New Trace dialog 452
NINF (negative infinity) 880
Noise figure measurement
Option R&S ZNB-K30 244
Non-operational data 1693
Notch (time gate) 365
NRFD 1705
NTR 890
NTRansition 892, 894
Number of Points 527, 537
Numerator 304
Numeric Editor (dialog) 63
Numeric Suffix (remote control) 912

O

OCXO Frequency Reference 26
Offset
Active 706
Offset Embed
All Offsets Off 707
All Offsets On 707
Balanced 710
Differential Match Embedding 713
Fixture Compensation 741
Ground Loop 711
Impedance Renormalization 712, 713
Loss 705
Loss active 706
Offset active 706
Port Pairs 748
Port Sets 708
Reset Offsets 705
Reset Offsets (and losses) 707
Single Ended deembedding 707
Single Ended embedding 707
Offset Embed dock widget 704
Balanced panel 710
Calculation Flow 705
Differential Match panel 713
Ground Loop panel 711
Impedance Renormalization panel 712
Offset 705
Overview 705
Port Sets panel 708
Single Ended panel 707
Offset Embed softtool 703
Balanced tab 751
Delta-L tab 771
Differential Match tab 757
Ground Loop tab 754
Offset tab 737
One Way Loss tab 743
Port Sets tab 748
Single Ended tab 745
Offset parameters 197
application 202
balanced ports 203
definition 198
Offset tab
Fixture Compensation dialog 741
On-screen keyboard
Analyzer 63
Windows 64
Open Recall... 780
Operating system 27
Operation complete 896
Remote 914
OPERation register 890, 897
Operational data 1693
Option
R&S ZN-B14 ..... 44, 260, 1707
R&S ZN-B15 ..... 260, 674, 1715
R&S ZN-Z15 44
R&S ZNB-B1 ..... 44, 258
R&S ZNB-B2 258
R&S ZNB-B4 ..... 26, 259
R&S ZNB-B10 ..... 44, 259
R&S ZNB-B12 ..... 44, 259
R&S ZNB-B19 262
R&S ZNB-B81 ..... 44, 264, 319
R&S ZNB-K2 ..... 218, 356
R&S ZNB-K3 ..... 231, 366
R&S ZNB-K4 233
R&S ZNB-K14 238
R&S ZNB-K17 243
R&S ZNB-K17 (Receiver Bandwidth 10 MHz) 1310
R&S ZNB-K19 244
R&S ZNB-K20 ..... 226, 374
R&S ZNB-K30 244
R&S ZNB-K50/R&S ZNB-K50P 248, 660
R&S ZNB4-B3n 264
R&S ZNB4-B52, R&S ZNB4-B54 263
R&S ZNB8-B3n 264
R&S ZNB8-B52, R&S ZNB8-B54 263
R&S ZNB40-B73/-U73 264
R&S ZNBT-K210 253
R&S ZNBT-K220 255
R&S ZNBT-K230 255
R&S ZNBT-K231 ..... 256, 257
R&S ZNBxx-B2 258
R&S ZNBxx-B22 262
R&S ZNBxx-B24 262
R&S ZVAB-B44 267
Options 828
Identification (remote) 914
Options (overview) 216
Options info 828
Output unit 886
Overlapped command execution 886

P

PAE ... 265
Parallel Poll ... 902
Parallel poll register enable
  Remote ... 915
Parameter (remote control) ... 912
Partial measurement ... 83
Password ... 31
  Service Function ... 839
PCIe interface ... 44
Peel ... 358
Permittivity ... 739
Persistent settings ... 79
Phase ... 439
Polar ... 440
Polar diagram ... 110
Port
  Gender ... 586, 622
  Offset ... 738
Port Config
  Define Parallel Measurement ... 694
  Port Settings ... 686
Port Config tab ... 685
Port Power Offset ... 651
Port settings (mixer) ... 327
Port Settings dialog ... 686
  Arb Frequency tab ... 686
  Arbitrary Power tab ... 690
  Cal Power Config dialog ... 691
  Frequency Result ... 687
  Modify Frequency Conversion dialog ... 688
  Power Conversion ... 691
  Power Result ... 690
  Receiver Level tab ... 692
  Slope ... 691
  Source Freq. Conversion ... 688
  Source Gen ... 687
  Source RF ... 687
Ports
  Calibration ... 567, 579, 583, 594, 663
Ports and Type
  SMARTerCal Wizard (Manual, MultiCal) ... 627
Power ... 531
Power (mixer) ... 329
Power (system settings) ... 820
Power added efficiency (dialog) ... 320
Power added efficiency (PAE) ... 265
Power Bw Avg softtool ... 531
  Average tab ... 534
  Bandwidth tab ... 533
  Receiver Power tab ... 532
  Source Power tab ... 531
Power Cal dialog
  Meas. Receiver mode ... 599
  Power mode ... 598
  Source Flatness mode ... 600
Power Cal dialog (Meas. Receiver) ... 599
  Port overview ... 599
  Start Cal Sweep ... 600
Power Cal dialog (Power) ... 598
  Start Cal Sweep ... 598
Power Cal dialog (Source Flatness) ... 600
  Start Cal Sweep ... 601
Power Cal wizard ... 595
  Calibration sweep diagram ... 596
  Screen elements ... 596
Power calibration ... 187
  measurement receiver ... 189
  Other Sources ... 649
  Power Meter ... 650
  Procedure ... 562
  Reference receiver ... 188
  Settings ... 648
  Source ... 188
Power Meter
  Auto Zero ... 650
  Power calibration ... 650
  SMARTerCal ... 605, 616, 620, 630
Power meter (external) ... 267
Power on and off ... 26
Power Port
  SMARTerCal ... 605, 616, 620
Power Sensor (menu) ... 318
Power Sensor tab ... 318
Power sweep ... 542
Power tab ... 528
PPC ... 1706
PPE ... 890, 896
PPU ... 1706
Precision frequency reference (OCXO)
  OptionR&S ZNB-B4 ... 259
Prepare Measurement of IM Order ... 340
Preset ... 904
Preset Configuration ... 812
Preset Scope ... 812
Preset vs. factory reset ... 811
Presets (system settings) ... 811
Print ... 784, 792
Printer ... 30
Printing data ... 76
Program example ... 1673
  Calibration ... 1673
Programming examples ... 1647
PTNA ... 194
PTOM ... 194
PTOSM ... 194
PTR ... 893
PTRansition ... 892
PTRL ... 194
PTRM ... 194
PTSM ... 194
Public folder ... 1254
PUOSM ... 194

Q

Quality factor (bandfilter) ... 102
Query ... 876
QUEStionable register ... 890, 897
  Summary bit ... 895
QUEStionable:INTegrity register ... 898
QUEStionable:LIMit register ... 897

R

R&S R&S ZNB-B12 ... 44
R&S ZN-B14 ... 44
R&S ZN-Z15 ... 44
R&S ZNB-B1 ... 44
R&S ZNB-B4 ... 26
R&S ZNB-B10 ... 44
R&S ZNB-B19 ... 43
R&S ZNB-B81 ... 44

R&S ZNB-K17 (Receiver Bandwidth 10 MHz) 1310
R&S ZVR compatible commands 1620
Ranges (DC inputs) 320
Ratio 127
Ratios tab 302
Read-only folder 1254
Real 441
Rear panel 41
USER PORT 1702
Recall Ripple Test 497
Recall set 80
Recall Set functions
Basic 779
Recall switch matrix configuration 781
Receiver bandwidth 10 MHz
Option R&S ZNB-K17 243
Receiver Bandwidth 10 MHz (option) 1310
Receiver Power tab 532
Receiver Step Attenuators
Options R&S ZNB4-B3n, R&S ZNB8-B3n 264
Receiving Port (intermodulation) 339
Recent Files 781
Reduced Through calibration 166
REF IN 43
Ref Mkr -> Mkr 508
REF OUT 43
Ref Pos 446
Ref Val = Marker 523
Ref Value 446
Reference impedance 119
Reference position 67
Reference Receiver Cal Power 652
Reference receiver calibration 188
Reference value 67
Reflection measurement (example) 77
Release notes 20
REM 1704
Remote control 868, 1656
Activation 872
Basic concepts 882
Introduction 868
Remote Control
Codec 810
Combine with manual control 875
Remote desktop 34
Remote Language 843
Remote Settings 842
Advanced 844
Define *IDN + *OPT... 843
GPIB Address 843
Instrument Messages 844
IP Address 842
Remote Language 843
Removable drive 43
Remove Last
SCPI recorder 906
REN 1705
Renormalization (of port impedances) 119
Renormalization theory
Power waves 302
Travelling waves 302
Replacing fuses 1698
Reset Directories 812
Reset Freq Conv Intermod 335
Reset Frequency Conversion 324
Reset Printer Page Setup 812
Reset values
Remote 915
Status reporting system 903
Resolution Enh(ancement) 360
Response (to queries) 876
Restart Manager dialog 561
Sweep Active Channel 562
Sweep All Channels 562
Restart Sweep 560
Restart Sweep on Std. Meas. 590, 626
RF Off All Channels 531
RFFE Config dialog 674
Sweep Sequencer tab 683
RFFE Config dialog (R&S ZN-B15/-Z15 Var. 03)
GPIOs tab 676
RFFE tab 675
Sweep Sequencer tab 676
RFFE interface
Configuration 677
external 44
Option R&S ZN-B15 ..... 260, 674, 1715
RFFE interface configuration 677
Apply changes (Set) 680
Basic 677
CLK 677
RFFE Command 677
SEND command 678
Seq. 677
VHigh 677
VIO 677
VLow 677
Voltage and current measurements 679
RFFE interface voltage and current measurements 679
Apply configuration changes (Set) 680
Output Voltage 679
Range 679
Results 680
Shunt 679
Start Meas 680
RFFE/GPIO interface
Sweep Sequencer 683
Ripple limit 135
files 143
Ripple Test 493, 495
Clear Test 495
Ranges 496
Ripple Check 494
Show Results All Traces 495
Show Ripple Limits 493
Rise Time 402
Rotary knob 39

S

S-parameter 117
multiport 118
S-parameter buttons 293
S-Parameter selector 293
S-Parameter Wizard 70, 294
S-Params tab 292
Safety instructions 15, 20
Same Color all Markers 805
Save (recall set) 781
Save As Recording
SCPI recorder 906
Save Recording
SCPI recorder 906

Save Ripple Test ... 497
Saving data ... 76
Scalar Mixer Mode ... 323
Scalar power calibration ... 187
Scale
  Scale Coupling ... 447
Scale Coupling tab ... 447
Scale softtool ... 444
  Scale Coupling tab ... 447
  Scale Values tab ... 444
  Zoom tab ... 447
Scale Values tab ... 444
Scale/Div ... 445
Scaling diagrams ... 65
Scan Instruments
  external generators ... 853
  external power meters ... 847
  External power meters ... 857
SCPI command structure ... 876
SCPI commands ... 876
SCPI recorder ... 905
  Clear All ... 907
  Export As ... 907
  Help ... 907
  List of recorded commands/script editor ... 906
  Load Recording ... 906
  Remove Last ... 906
  Save As Recording ... 906
  Save Recording ... 906
  Start Recording ... 905
SCPI status register ... 892
  Event status register ... 896
  IST flag ... 896
  Parallel poll enable register ... 896
  Service request enable register ... 895
  Status byte ... 895
  STATus:OPERation register ... 897
  STATus:QUEStionable register ... 897
  STATus:QUEStionable:INTegrity ... 898
  STATus:QUEStionable:LIMit ... 897
  Structure ... 890
Screen elements ... 93
Screen Keyboard ... 795
Screen saver ... 36
SDC ... 1706
Search functions ... 101
Search Path for additional Cal Kits and Connector Types ... 814
Search Range dialog ... 511
  Range Limit Lines On ... 512
  Search Range ... 511
  Select Marker ... 511
Second internal generator
  Option R&S ZNB-B2 ... 258
  Option R&S ZNBxx-B2 ... 258
Security procedures ... 20
Seg X-Axis ... 544
Segmented AGC ... 701
Segmented sweep ... 541
  Define Segments dialog ... 545
  Export segments ... 548
  Import segments ... 548
Select Parameter dialog ... 482
Select Ports dialog ... 482
Self-test
  Remote ... 916
Selftest ... 830
Selftest info ... 830
Sequential command execution ... 886
Serial poll ... 902
Service
  Recalibration ... 1698
Service Function ... 838, 839
  Password ... 839
Service Function dialog ... 838
Service manual ... 19
Service request ... 901
Service request (SRQ) ... 868
Service Request Enable ... 890, 895
Service request enable register (SRE)
  Remote ... 915
Set by Marker ... 522
  Center ... 523
  Max ... 523
  Min ... 523
  Ref Value ... 523
  Span ... 523
  Start ... 523
  Stop ... 523
  Zero Delay at Marker ... 523
Set to Default Port Assignment ... 572, 608
Setup
  Define Physical Ports ... 864
  Device LAN Configuration ... 858
  External Generators ... 851
  External Matrices ... 855
  External Power Meter Config ... 849
  External Power Meters ... 845
  Reference ... 841
  Switch Matrix RF Connections ... 859
Setup info ... 828
Setup information ... 828
Setup softtool ... 809
  Freq. Ref. tab ... 841
  Remote Settings tab ... 842
  Setup tab ... 810
SFD
  Advanced settings ... 724
Shape (time gate) ... 365
Shift JIS ... 810
Shift trace
  response ... 473
  stimulus ... 473
Shift Trace ... 473
Show All Data ... 459
Show All Mem ... 459
Show All Traces ... 452
Show as ... 308
Show Cal Kit Label ... 814
Show Limit Fail Symbols ... 804
Show Point List ... 547
Show/Hide data trace ... 457
Show/Hide memory trace ... 457
Side lobe level
  Frequency window ... 359
Side Lobe Level (time gate) ... 365
Single sweep ... 559
Single sweep mode
  Sweeps (# of) ... 559
Single-ended mode ... 129, 131
Skew ... 403
Sliding Match ... 172

Smart fixture de-embedding
R&S ZNB-K230 255
Smarter calibration 193
SMARTerCal 193
Ports 604, 615, 619, 629
Ports, Standards and Power 625
Power Meter 605, 616, 620, 630
Power Port 605, 616, 620
Power Sweep Diagram 625
Procedure 562
Type 620, 629
SMARTerCal (Manual) Wizard
Calibration 623
SMARTerCal wizard
Cal Unit 601
SMARTerCal wizard (Cal Unit, MultiCal)
Connections 616
Define Calibration dialog 613
MultiCal 612
Ports 613
SMARTerCal wizard (Cal Unit) 601
Abort Sweep 610
Apply 611
Connections 606
Data acquisition 609
Ports 602
Start Cal Sweep 610
SMARTerCal Wizard (Manual, MultiCal)
Connectors and Cal Kits 630
Ports and Type 627
SMARTerCal wizard (manual)
Connectors and Cal Kits 620
Ports and Type 618
SMARTerCal wizard (MultiCal)
Cal Unit 612
Smith 439
Smith chart 111
Smoothing 472
Aperture 472
Softtool panel
Minimize/maximize 809
Softtools 51
Software Option Info 829
Source
Calibration 584, 594
Source flatness calibration 188
Source Port 305
Source Power tab 531
Span = Marker 523
Span Frequency 527
SPD 1706
SPE 1706
Specifications document 20
Split All 798
Split diagrams 799
Dual Split 799
Quad Split 799
Triple Split 799
Split Type 800
Src. for Rcvr. Cal 339
SRE 890, 895
SRQ 890, 901, 1705
Stability (softkey) 317
Stability factors 134
Stability tab 317
Standby and ready state 26
Standby key 40
Start = Marker 523
Start Frequency 527
Start Power 532
Start Recording
SCPI recorder 905
Startup and shutdown 26
Status bar 53
Status Bar
Show/hide 808
Status byte
Remote 913, 916
Status registers 890
Status reporting system
common commands 913
Common commands 911
Error queue 903
Parallel poll 902
Queries 903
Reset values 903
Serial poll 902
Service request 901
STB 890, 895
Step attenuators 532
Step Size (dialog) 63
Stimulus Axis 528
Stimulus keys
[Center] 38
[Span] 38
[Start] 38
[Stop] 38
Stimulus softtool 526
Power tab 528
Source Power tab 531
Stimulus tab 526
Time Domain tab 529
Stimulus tab 526
Center Frequency 527
CW Frequency 528
Number of Points 527
Power 531
Span Frequency 527
Start Frequency 527
Stop Frequency 527
Stop = Marker 523
Stop Frequency 527
Stop Power 532
String 880
Support 1691
Sweep
Sweep Params 536
Sweep Active Channel 562
Sweep All Channels 562
Sweep Control
Restart Manager dialog 561
Sweep Control tab 558
Sweep History 1670
Sweep mode
Continuous 559
Single 559
Sweep range selection 72
Sweep range setting 66
Sweep Sequencer (RFFE/GPIO interface) 683
Sweep settings 536
Sweep softtool
Sweep Control tab 558
Sweep Type tab 539

Trigger In tab 552
Trigger Out tab 557
Sweep Time 538
Sweep Type
CW Mode 542
Lin Freq 540
Log Freq 540
Power 542
Segmented 541
Time 543
Sweep Type tab 539
Sweeps (# of) 559
Switch Matrices
Add Switch Matrix (dialog) 860
Delete Switch Matrix (dialog) 861
Switch Matrix RF Connections dialog 859
SWR 440
Syntax elements (SCPI, overview) 880
System Config dialog 811
Advanced tab 818
Calibration tab 813
Channel Bits tab 817
Power tab 820
Presets tab 811
User Interface tab 815
System drive 43
System keys
[Applic] 38
[Display] 38
[File Print] 38
[File] 38
[Help] 38
[Preset] 38
[Print] 38
[Setup] 38
Windows 38
System settings 809

T

Target Search 516, 517
Search Left 517
Target Format 517
Target Value 516
Task bar
Show/hide 808
TDR application
TDR Setup tab 375
TDR Stimulus Settings 378
TDR Wizard 376
TDR Application
Eye Diag tab 381
Eye Diagram dialog 383
Eye Mask Test 397
Rise Time 402
Time Gate 405
TDR measurements 374
Skew 403
X Axis 409
Y Axis 409
TDR Setup
TDR Stimulus 378
TDR Wizard 376
TDR Stimulus - Advanced Settings dialog 380
TDR Stimulus Settings 378
Advanced 380
Basic 379
TDR Wizard 376
Test port 40
Text parameter 880
Time Center 529
Time Domain 357
Time domain analysis 363
Option R&S ZNB-K2 ..... 218, 356
Time domain reflectometry 226
Time domain signal integrity tests 226
Time Domain tab 356, 529
Time Domain Transform 218
Time domain transformation
Impulse Response 359
Type 359
Time Gate 364, 405
Bandpass 365
Notch 365
Shape 365
Show Range Lines 364
Side Lobe Level 365
Time Gate tab 363
Time Span 529
Time Start 529
Time Stop 529
Time sweep 543
Time vs. Distance 530
Title (diagram) 94
Title bar 49
Show/hide 808
Title Bar Task Bar 795
Tone Distance (intermodulation) 340
Tool 3 ... Tool 8 795
Tool Bar
Show/hide 808
Toolbar 49
Touchscreen 36
Touchscreen (R&S ZNB only)
Configuration 809
Enabled 809
Lock Diagrams 809
Lock Screen 809
Touchstone (*.s<n>p) file 144
Trace 57, 80, 94
Active memory trace 451
Dflt Marker Frmt 443
Diagram 451
Move 450
programming examples 1650
Show/hide 451
Trace Colors per Diagram 805
Trace Config
Hold 473
Max Hold 473
Min Hold 473
Shift Trace 473
Smoothing 472
Trace Config softtool 449
All Mem & All Data tab 458
Hold 471
Mem tab 455
Shift Trace 471
Smooth & Shift & Hold tab 471
Smoothing 471
Time Gate tab 363
Trace Data tab 475

Trace Statistics tab ... 465
Traces tab ... 449
Trace Data ... 787
Export ... 475
Import ... 475
Import Complex Data ... 476
Trace Data import
Auto Distribute traces ... 482
Deselect All traces ... 482
Select All traces ... 482
Select Parameter ... 482
Trace files ... 143
Trace format ... 438
dB Mag ... 438
Delay ... 442
Imag ... 442
Inv Smith ... 442
Lin Mag ... 441
Log Mag ... 441
Phase ... 439
Polar ... 440
Real ... 441
Smith ... 439
SWR ... 440
Unwr Phase ... 440
Trace formats ... 108
Trace Info ... 802
Trace keys
[Format] ... 37
[Line] ... 37
[Marker] ... 37
[Meas] ... 37
[Scale] ... 37
[Trace Config] ... 37
Trace list ... 95
Trace Manager dialog ... 453
Add ... 454
Couple All Channels ... 455
Couple All Scales ... 455
Decouple All Channels ... 455
Decouple All Scales ... 455
Delete ... 455
Trace Math ... 460
Data - <Mem> ... 461
Data / <Mem> ... 461
User Defined ... 461
Trace Statistics ... 465
Compr. Point ... 468
Compr. Val. ... 468
Evaluation Range ... 470
Evaluation Range Limit Lines On ... 471
Flatness/Gain/Slope ... 467
Format ... 467
Mean/Std Dev/RMS ... 466
Min/Max/Peak-Peak ... 466
Phase/El Length ... 467
Tracking
Marker search, target search ... 510
Transmission measurement (example) ... 71
Trigger
Delay ... 555
Event (remote) ... 916
External ... 553
Manual ... 553, 554
Multiple ... 553
None ... 553
Sequence ... 554
Signal Type ... 555
Source ... 553
Trigger In ... 1703
Trigger Out ... 1703
Trigger In ... 43, 1703
Trigger In tab ... 552
Controls ... 553
Trigger Manager dialog ... 555
Source ... 556
Trigger Out ... 43, 1703
Trigger Out tab ... 557
Troubleshooting ... 1687
TTL Pass ... 820
Two Port Configuration dialog ... 654
Power loss list ... 654
Transm. Coefficients ... 654
Typical (cal kit) ... 171

U

Unbalance-balance conversion ... 129
Universal command ... 876, 1706
Universal interface
Option R&S ZN-B14 ... 260, 1707
Universal Interface ... 1707
Unwr Phase ... 440
UP ... 880
Upper edge (bandfilter) ... 102
Upper limit ... 135
Upper Tone (intermodulation) ... 339
Upper/lower case (remote control) ... 912
USB
front panel ... 41
USB Device
rear panel ... 43
USB Host
rear panel ... 43
USB-to-GPIB Adapter
Option R&S ZVAB-B44 ... 267
USB-to-IEC/IEEE adapter
Option R&S ZVAB-B44 ... 267
Use Cal tab
All Power Cals Off ... 656
All Power Cals On ... 656
Controls ... 655
Recall Last Cal Set ... 656
User Cal Active ... 655
Use Trc Color for Limit Lines ... 805
User Def Math dialog ... 461
Expression builder ... 462
Result is Wave Quantity ... 464
User Interface (system settings) ... 815
Conductance in Embedding Networks ... 816
Decimal Places ... 816
Reset Colors ... 817
Reset Decimal Places ... 817
Reset Dialogs ... 817
Reset Units Prefix ... 817
Show Sweep Symbols ... 815
Transparent Info Fields ... 815
Units Prefix ... 817
Use Default Tab for Hardkey ... 816
USER PORT ... 43, 1702
TTL Pass output ... 820
UTF-8 ... 810

V

Velocity Factor 739
View / Modify Cal Kit Standards dialog 641, 642
  Characteristic impedance Z0 642
  Delay 642
  Electrical Length 642
  Load parameters 642
  Loss 642
  Offset Loss 642
  Offset parameters 642
Virtual differential mode 129
Virus protection 31
VISA 868
VNA Port Assignment 647
VXI-11 868

W

Wait
  Remote 916
Wave quantities
  Source port 308
Wave quantity 126, 307
Wave tab 305
White papers 20
Window mode 808
Windows 27
Windows Explorer 795
Wizard 70

Y

Y <- (softkey) 311
Y <- S-Parameters
  Model 312
Y-parameter 125
Y-parameters 312
Y-Z-Params tab 312
Y/Z-Parameter (softkey) 313
Y←Sij tab 311
Y<-S11 (softkey) 311
Y<-S12 (softkey) 311
Y<-S21 (softkey) 311
Y<-S22 (softkey) 311
Y11 (softkey) 313
Y12 (softkey) 313
Y21 (softkey) 313
Y22 (softkey) 313

Z

Z ← (softkey) 309
Z <- S-Parameters
  Model 310
Z-parameter 124
Z-parameters 312
Z←S11 (softkey) 309
Z←S12 (softkey) 309
Z←S21 (softkey) 309
Z←S22 (softkey) 309
Z←Sij tab 308
Z11 (softkey) 314
Z12 (softkey) 314
Z21 (softkey) 314
Z22 (softkey) 314
Zero Delay at Marker 523
Zoom 65
  Max 449
  Min 449
  Overview On 449
  Start 449
  Stop 449
Zoom Reset 449
Zoom Select 448
Zoom tab 447