# Operating Manual

# Vector Network Analyzers

## R&S® ZVA 8 / ZVA 24 / ZVA 40

**1145.1110.08/10 / 24/26 / 40**

## R&S® ZVB 4 / ZVB 8 / ZVB 14 / ZVB 20

**1145.1010.04/05/06 / 08/09/10 / 14/17 / 20/22/23/25**

## R&S® ZVT 8 / ZVT 20

**1300.0000.08/20**

Printed in Germany

ROHDE&SCHWARZ
Test and Measurement Division

R&S® is a registered trademark of Rohde & Schwarz GmbH & Co. KG.
Trade names are trademarks of the owners.

This product includes software developed by the OpenSSL Project for use in the
OpenSSL Toolkit (http://www.openssl.org).

# Tabbed Divider Overview

**R&S® ZVA/ZVB/ZVT Documentation Map**

**What's New in this Revision**

## Data Sheet

**Safety Instructions**
**Certificate of Quality**
**EU Certificate of Conformity**
**List of R&S Representatives**

**Glossary of Terms**

**Tabbed Divider**

| | | |
|---|---|---|
| **1** | **Chapter 1:** | **Preparing for Use: R&S ZVA**<br>**Preparing for Use: R&S ZVB**<br>**Preparing for Use: R&S ZVT** |
| **2** | **Chapter 2:** | **Getting Started** |
| **3** | **Chapter 3:** | **System Overview** |
| **4** | **Chapter 4:** | **GUI Reference** |
| **5** | **Chapter 5:** | **Remote Control** |
| **6** | **Chapter 6:** | **Command Reference** |
| **7** | **Chapter 7:** | **Program Examples** |
| **8** | **Chapter 8:** | **Maintenance, Error Messages,Interfaces** |
| **9** | **Chapter 9:** | **Measurement Examples** |
| **10** | **Index** | |

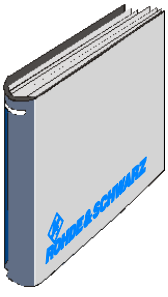
# R&S® ZVA/ZVB/ZVT Documentation Map

## Help System

The help system is embedded in the instrument, offering quick, context-sensitive reference to the information needed for operation and programming. The help contains the complete user documentation for the network analyzer.

Three separate help files ZVA.chm, ZVB.chm, and ZVT.chm are provided for the network analyzers R&S ZVA, R&S ZVB, and R&S ZVT, respectively. You can also transfer the help files to your PC and use them as a standalone help.

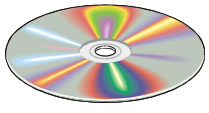
## Quick Start Guide



The quick start guide contains the data sheet, describes everything that is needed to put the instrument into operation and helps you get familiar with the analyzer. The quick start guide gives an introduction to the instrument's functionality and provides procedures for typical measurement tasks.

Three separate quick start guides are provided for the network analyzers R&S ZVA, R&S ZVB, and R&S ZVT, respectively.

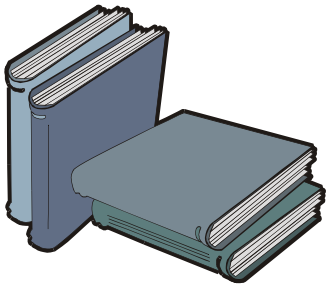
## Documentation CD-ROM



The CD-ROM provides the complete user documentation for the network analyzer family:

- The online help systems for R&S ZVA, R&S ZVB, and R&S ZVT in two different HTML-based formats (*.chm for transfer to the hard disk and WebHelp for viewing from the CD).
- A printable (*.pdf) version of the online help.
- The quick start manuals in printable form.
- The data sheets in printable form.
- Links to different useful sites in the R&S internet.

## Optional Documentation



The present printable/printed version of the help system provides the contents of the quick start manuals plus the complete reference information for operation and programming contained in the help systems. This manual can be ordered as an option (stock no. 1155.1084.12); see ordering information in the data sheet.

The manual is valid for all analyzer types R&S ZVA, R&S ZVB, and R&S ZVT. Chapter 1 is divided in three parts for the three analyzer types. The remaining chapters are based on a R&S ZVA, which is the instrument with the most comprehensive functionality. Most of the features are also available on a R&S ZVB and on a R&S ZVT that is controlled from the Graphical User Interface displayed on an external monitor.

# What's New in this Revision…

This manual describes version V2.20 of the ZVA/ZVB/ZVT firmware. Compared to previous versions, the new firmware provides numerous extensions and improvements. The history of the firmware depends on your analyzer type, it is documented at the beginning of the help system, section *Welcome – What's New?*.

To check your ZVA/ZVB/ZVT firmware version, click *Help – About Nwa...*

ROHDE&SCHWARZ

EC Certificate of Conformity

CE

Certificate No.: 2006-23

This is to certify that:

| Equipment type | Stock No. | Designation |
|---|---|---|
| ZVA8 | 1145.1110.08/.10 | Vector Network Analyzer |
| ZVA8-B16 | 1164.0209.08/.10 | Direct Generator/Receiver Access |
| ZVAB-B18 | 1164.0715.02 | Removable FLASH Disk |
| ZVAB-B19 | 1164.1111.02 | Second Removable FLASH Disk |
| ZVA8-B21 | 1164.0009.02 | Generator Attenuator Port 1 |
| ZVA8-B22 | 1164.0015.02 | Generator Attenuator Port 2 |
| ZVA8-B23 | 1164.0021.02 | Generator Attenuator Port 3 |
| ZVA8-B24 | 1164.0038.02 | Generator Attenuator Port 4 |
| ZVA8-B31 | 1164.0044.02 | Receiver Attenuator Port 1 |
| ZVA8-B32 | 1164.0050.02 | Receiver Attenuator Port 2 |
| ZVA8-B33 | 1164.0067.02 | Receiver Attenuator Port 3 |
| ZVA8-B34 | 1164.0073.02 | Receiver Attenuator Port 4 |

complies with the provisions of the Directive of the Council of the European Union on the approximation of the laws of the Member States

- relating to electrical equipment for use within defined voltage limits (2006/95/EC)
- relating to electromagnetic compatibility (89/336/EEC revised by 91/263/EEC, 92/31/EEC, 93/68/EEC)

Conformity is proven by compliance with the following standards:

EN 61010-1 : 2001
EN 61326 : 1997 + A1 : 1998 + A2 : 2001 + A3 : 2003
EN 55011 : 1998 + A1 : 1999 + A2 : 2002, Class A
EN 61000-3-2 : 2000 + A2 : 2005
EN 61000-3-3 : 1995 + A1 : 2001

For the assessment of electromagnetic compatibility, the limits of radio interference for Class A equipment as well as the immunity to interference for operation in industry have been used as a basis.

Affixing the EC conformity mark as from 2006

**ROHDE & SCHWARZ GmbH & Co. KG**
**Mühldorfstr. 15, D-81671 München**

Munich, 2007-05-30

Central Quality Management MF-QZ / Radde

ROHDE&SCHWARZ

EC Certificate of Conformity

CE

Certificate No.: 2007-18

This is to certify that:

| Equipment type | Stock No. | Designation |
|---|---|---|
| ZVA24 | 1145.1110.24/.26 | Vector Network Analyzer |
| ZVA24-B16 | 1164.0209.24/.26 | Direct Generator/Receiver Access |
| ZVAB-B18 | 1164.0715.02 | Removable FLASH Disk |
| ZVAB-B19 | 1164.1111.02 | Second Removable FLASH Disk |
| ZVA24-B21 | 1164.0109.02 | Generator Attenuator Port 1 |
| ZVA24-B22 | 1164.0115.02 | Generator Attenuator Port 2 |
| ZVA24-B23 | 1164.0121.02 | Generator Attenuator Port 3 |
| ZVA24-B24 | 1164.0138.02 | Generator Attenuator Port 4 |
| ZVA24-B31 | 1164.0144.02 | Receiver Attenuator Port 1 |
| ZVA24-B32 | 1164.0150.02 | Receiver Attenuator Port 2 |
| ZVA24-B33 | 1164.0167.02 | Receiver Attenuator Port 3 |
| ZVA24-B34 | 1164.0173.02 | Receiver Attenuator Port 4 |

complies with the provisions of the Directive of the Council of the European Union on the approximation of the laws of the Member States

- relating to electrical equipment for use within defined voltage limits (2006/95/EC)
- relating to electromagnetic compatibility (89/336/EEC revised by 91/263/EEC, 92/31/EEC, 93/68/EEC)

Conformity is proven by compliance with the following standards:

EN 61010-1 : 2001
EN 61326 : 1997 + A1 : 1998 + A2 : 2001 + A3 : 2003
EN 55011 : 1998 + A1 : 1999 + A2 : 2002, Class A
EN 61000-3-2 : 2000 + A2 : 2005
EN 61000-3-3 : 1995 + A1 : 2001

For the assessment of electromagnetic compatibility, the limits of radio interference for Class A equipment as well as the immunity to interference for operation in industry have been used as a basis.

Affixing the EC conformity mark as from 2007

**ROHDE & SCHWARZ GmbH & Co. KG**
**Mühldorfstr. 15, D-81671 München**

Munich, 2007-05-30

Central Quality Management MF-QZ / Radde

ROHDE&SCHWARZ

EC Certificate of Conformity

CE

Certificate No.: 2006-26

This is to certify that:

| Equipment type | Stock No. | Designation |
|---|---|---|
| ZVA40 | 1145.1110.40/.42 | Vector Network Analyzer |
| ZVA40-B16 | 1164.0209.40/.42 | Direct Generator/Receiver Access |
| ZVAB-B18 | 1164.0715.02 | Removable FLASH Disk |
| ZVAB-B19 | 1164.1111.02 | Second Removable FLASH Disk |
| ZVA40-B21 | 1302.5409.02 | Generator Attenuator Port 1 |
| ZVA40-B22 | 1302.5419.02 | Generator Attenuator Port 2 |
| ZVA40-B23 | 1302.5421.02 | Generator Attenuator Port 3 |
| ZVA40-B24 | 1302.5438.02 | Generator Attenuator Port 4 |
| ZVA40-B31 | 1302.5444.02 | Receiver Attenuator Port 1 |
| ZVA40-B32 | 1302.5450.02 | Receiver Attenuator Port 2 |
| ZVA40-B33 | 1302.5467.02 | Receiver Attenuator Port 3 |
| ZVA40-B34 | 1302.5473.02 | Receiver Attenuator Port 4 |

complies with the provisions of the Directive of the Council of the European Union on the approximation of the laws of the Member States

- relating to electrical equipment for use within defined voltage limits (2006/95/EC)

- relating to electromagnetic compatibility (89/336/EEC revised by 91/263/EEC, 92/31/EEC, 93/68/EEC)

Conformity is proven by compliance with the following standards:

EN 61010-1 : 2001
EN 61326 : 1997 + A1 : 1998 + A2 : 2001 + A3 : 2003
EN 55011 : 1998 + A1 : 1999 + A2 : 2002, Class A
EN 61000-3-2 : 2000 + A2 : 2005
EN 61000-3-3 : 1995 + A1 : 2001

For the assessment of electromagnetic compatibility, the limits of radio interference for Class A equipment as well as the immunity to interference for operation in industry have been used as a basis.

Affixing the EC conformity mark as from 2006

**ROHDE & SCHWARZ GmbH & Co. KG**
**Mühldorfstr. 15, D-81671 München**

Munich, 2007-05-30

Central Quality Management MF-QZ / Radde

ROHDE&SCHWARZ

EC Certificate of Conformity

CE

Certificate No.: 2007-19

This is to certify that:

| Equipment type | Stock No. | Designation |
|---|---|---|
| ZVB4 | 1145.1010.04/.05/.06 | Vector Network Analyzer |
| ZVB8 | 1145.1010.08/.09/.10 | |
| ZVB14 | 1145.1010.14/.17/.19 | |
| ZVB20 | 1145.1010.20/.22/.23/.25 | |
| ZVB14-B16 | 1164.1240.17/.19 | Direct Generator/Receiver Access |
| ZVB20-B16 | 1164.1240.23/.25 | Direct Generator/Receiver Access |
| ZVAB-B18 | 1164.0715.02 | Removable FLASH Disk |
| ZVAB-B19 | 1164.1111.02 | Second Removable FLASH Disk |
| ZVB8-B21 | 1302.5480.02 | Generator Attenuator Port 1 |
| ZVB8-B22 | 1302.5073.02 | Generator Attenuator Port 2 |
| ZVB8-B23 | 1302.5496.02 | Generator Attenuator Port 3 |

complies with the provisions of the Directive of the Council of the European Union on the approximation of the laws of the Member States

- relating to electrical equipment for use within defined voltage limits (2006/95/EC)
- relating to electromagnetic compatibility (89/336/EEC revised by 91/263/EEC, 92/31/EEC, 93/68/EEC)

Conformity is proven by compliance with the following standards:

EN 61010-1 : 2001
EN 61326 : 1997 + A1 : 1998 + A2 : 2001 + A3 : 2003
EN 55011 : 1998 + A1 : 1999 + A2 : 2002, Class A
EN 61000-3-2 : 2000 + A2 : 2005
EN 61000-3-3 : 1995 + A1 : 2001

For the assessment of electromagnetic compatibility, the limits of radio interference for Class A equipment as well as the immunity to interference for operation in industry have been used as a basis.

Affixing the EC conformity mark as from 2007

**ROHDE & SCHWARZ GmbH & Co. KG**
**Mühldorfstr. 15, D-81671 München**

Munich, 2007-05-30

Central Quality Management MF-QZ / Radde

ROHDE&SCHWARZ

EC Certificate of Conformity

CE

Certificate No.: 2007-20

This is to certify that:

| Equipment type | Stock No. | Designation |
|---|---|---|
| ZVT8 | 1300.0000.08 | Network Analyzer |
| ZVT8-B16 | 1300.1706.11 - .18 | Direct Generator/Receiver Access Port 1 - 8 |
| ZVT8-B63 | 1300.1506.03/.13 | Additional Port 3 |
| ZVT8-B64 | 1300.1506.04/.14 | Additional Port 4 |
| ZVT8-B65 | 1300.1506.05/.15 | Additional Port 5 |
| ZVT8-B66 | 1300.1506.06/.16 | Additional Port 6 |
| ZVT8-B67 | 1300.1506.07/.17 | Additional Port 7 |
| ZVT8-B68 | 1300.1506.08/.18 | Additional Port 8 |

complies with the provisions of the Directive of the Council of the European Union on the approximation of the laws of the Member States

- relating to electrical equipment for use within defined voltage limits (2006/95/EC)
- relating to electromagnetic compatibility (89/336/EEC revised by 91/263/EEC, 92/31/EEC, 93/68/EEC)

Conformity is proven by compliance with the following standards:

EN 61010-1 : 2001
EN 61326 : 1997 + A1 : 1998 + A2 : 2001 + A3 : 2003
EN 55011 : 1998 + A1 : 1999 + A2 : 2002, Class A
EN 61000-3-2 : 2000 + A2 : 2005
EN 61000-3-3 : 1995 + A1 : 2001

For the assessment of electromagnetic compatibility, the limits of radio interference for Class A equipment as well as the immunity to interference for operation in industry have been used as a basis.

Affixing the EC conformity mark as from 2007

**ROHDE & SCHWARZ GmbH & Co. KG**
**Mühldorfstr. 15, D-81671 München**

Munich, 2007-05-31

Central Quality Management MF-QZ / Radde

ROHDE&SCHWARZ

EC Certificate of Conformity

CE

Certificate No.: 2007-21

This is to certify that:

| Equipment type | Stock No. | Designation |
|---|---|---|
| ZVT20 | 1300.0000.20 | Network Analyzer |
| ZVT20-B16 | 1300.1635.11 - .16 | Direct Generator/Receiver Access Port 1 - 8 |
| ZVT20-B63 | 1300.1606.03 | Additional Port 3 |
| ZVT20-B64 | 1300.1606.04 | Additional Port 4 |
| ZVT20-B65 | 1300.1606.05 | Additional Port 5 |
| ZVT20-B66 | 1300.1606.06 | Additional Port 6 |

complies with the provisions of the Directive of the Council of the European Union on the approximation of the laws of the Member States

- relating to electrical equipment for use within defined voltage limits (2006/95/EC)
- relating to electromagnetic compatibility (89/336/EEC revised by 91/263/EEC, 92/31/EEC, 93/68/EEC)

Conformity is proven by compliance with the following standards:

EN 61010-1 : 2001
EN 61326 : 1997 + A1 : 1998 + A2 : 2001 + A3 : 2003
EN 55011 : 1998 + A1 : 1999 + A2 : 2002, Class A
EN 61000-3-2 : 2000 + A2 : 2005
EN 61000-3-3 : 1995 + A1 : 2001

For the assessment of electromagnetic compatibility, the limits of radio interference for Class A equipment as well as the immunity to interference for operation in industry have been used as a basis.

Affixing the EC conformity mark as from 2007

**ROHDE & SCHWARZ GmbH & Co. KG**
**Mühldorfstr. 15, D-81671 München**

Munich, 2007-05-31

Central Quality Management MF-QZ / Radde

ROHDE&SCHWARZ

EC Certificate of Conformity

CE

Certificate No.: 2005-11

This is to certify that:

| Equipment type | Stock No. | Designation |
|---|---|---|
| ZV-Z51 | 1164.0515.30/.70 | Calibration Unit |
| ZV-Z52 | 1164.0521.30/.72 | Calibration Unit |
| ZV-Z58 | 1164.0638.38/.78 | Calibration Unit |

complies with the provisions of the Directive of the Council of the European Union on the approximation of the laws of the Member States

- relating to electrical equipment for use within defined voltage limits (73/23/EEC revised by 93/68/EEC)
- relating to electromagnetic compatibility (89/336/EEC revised by 91/263/EEC, 92/31/EEC, 93/68/EEC)

Conformity is proven by compliance with the following standards:

EN61010-1 : 2001
EN55011 : 1998 + A1 : 1999 + A2 : 2002, Class B
EN61326 : 1997 + A1 : 1998 + A2 : 2001 + A3 : 2003

For the assessment of electromagnetic compatibility, the limits of radio interference for Class B equipment as well as the immunity to interference for operation in industry have been used as a basis.

Affixing the EC conformity mark as from 2005

**ROHDE & SCHWARZ GmbH & Co. KG**
**Mühldorfstr. 15, D-81671 München**

Munich, 2006-05-23

Central Quality Management MF-QZ / Radde

**Before putting the product into operation for the first time, make sure to read the following**

# Safety Instructions

All plants and locations of the Rohde & Schwarz group of companies make every effort to keep the safety standard of our products up to date and to offer our customers the highest possible degree of safety. Our products and the auxiliary equipment required for them are designed and tested in accordance with the relevant safety standards. Compliance with these standards is continuously monitored by our quality assurance system. The product described here has been designed and tested in accordance with the EC Certificate of Conformity and has left the manufacturer's plant in a condition fully complying with safety standards. To maintain this condition and to ensure safe operation, observe all instructions and warnings provided in this manual. If you have any questions regarding these safety instructions, the Rohde & Schwarz group of companies will be happy to answer them.

Furthermore, it is your responsibility to use the product in an appropriate manner. This product is designed for use solely in industrial and laboratory environments or in the field and must not be used in any way that may cause personal injury or property damage. You are responsible if the product is used for an intention other than its designated purpose or in disregard of the manufacturer's instructions. The manufacturer shall assume no responsibility for such use of the product.

The product is used for its designated purpose if it is used in accordance with its product documentation and within its performance limits (see data sheet, documentation, the following safety instructions). Using the product requires technical skills and a basic knowledge of English. It is therefore essential that the product be used exclusively by skilled and specialized staff or thoroughly trained personnel with the required skills. If personal safety gear is required for using Rohde & Schwarz products, this will be indicated at the appropriate place in the product documentation.

## Symbols and safety labels

| | | | | | | | |
|---|---|---|---|---|---|---|---|
| | | | | | | | |
| Observe product documentation | Weight indication for units >18 kg | Danger of electric shock | Warning! Hot surface | PE terminal | Ground | Ground terminal | Attention! Electrostatic sensitive devices |

| | | | | | |
|---|---|---|---|---|---|
| | | | | | |
| Supply voltage ON/OFF | Standby indication | Direct current (DC) | Alternating current (AC) | Direct/alternating current (DC/AC) | Device fully protected by double/reinforced insulation |

Observing the safety instructions will help prevent personal injury or damage of any kind caused by dangerous situations. Therefore, carefully read through and adhere to the following safety instructions before putting the product into operation. It is also absolutely essential to observe the additional safety instructions on personal safety that appear in relevant parts of the product documentation. In these safety instructions, the word "product" refers to all merchandise sold and distributed by the Rohde & Schwarz group of companies, including instruments, systems and all accessories.

## Tags and their meaning

| | |
|---|---|
| DANGER | This tag indicates a definite hazard carrying a high risk of death or serious injury if not avoided. |
| WARNING | This tag indicates a possible hazard carrying a medium risk of death or (serious) injury if not avoided. |
| CAUTION | This tag indicates a hazard carrying a low risk of minor or moderate injury if not avoided. |
| ATTENTION | This tag indicates the possibility of incorrect use that can cause damage to the product. |
| NOTE | This tag indicates a situation where the user should pay special attention to operating the product but which does not lead to damage. |

These tags are in accordance with the standard definition for civil applications in the European Economic Area. Definitions that deviate from the standard definition may also exist in other economic areas or military applications. It is therefore essential to make sure that the tags described here are always used only in connection with the related product documentation and the related product. The use of tags in connection with unrelated products or documentation can result in misinterpretation and thus contribute to personal injury or material damage.

## Basic safety instructions

1. The product may be operated only under the operating conditions and in the positions specified by the manufacturer. Its ventilation must not be obstructed during operation. Unless otherwise specified, the following requirements apply to Rohde & Schwarz products:
   prescribed operating position is always with the housing floor facing down, IP protection 2X, pollution severity 2, overvoltage category 2, use only in enclosed spaces, max. operation altitude 2000 m above sea level, max. transport altitude 4500 m above sea level.
   Unless specified otherwise in the data sheet, a tolerance of ±10% shall apply to the nominal voltage and of ±5% to the nominal frequency.
2. Applicable local or national safety regulations and rules for the prevention of accidents must be observed in all work performed. The product may be opened only by authorized, specially trained personnel. Prior to performing any work on the product or opening the product, the product must be disconnected from the supply network. Any adjustments, replacements of parts, maintenance or repair must be carried out only by technical personnel authorized by Rohde & Schwarz. Only original parts may be used for replacing parts relevant to safety (e.g. power switches, power transformers, fuses). A safety test must always be performed after parts relevant to safety have been replaced (visual inspection, PE conductor test, insulation resistance measurement, leakage current measurement, functional test).
3. As with all industrially manufactured goods, the use of substances that induce an allergic reaction (allergens, e.g. nickel) such as aluminum cannot be generally excluded. If you develop an allergic reaction (such as a skin rash, frequent sneezing, red eyes or respiratory difficulties), consult a physician immediately to determine the cause.

4. If products/components are mechanically and/or thermically processed in a manner that goes beyond their intended use, hazardous substances (heavy-metal dust such as lead, beryllium, nickel) may be released. For this reason, the product may only be disassembled, e.g. for disposal purposes, by specially trained personnel. Improper disassembly may be hazardous to your health. National waste disposal regulations must be observed.
5. If handling the product yields hazardous substances or fuels that must be disposed of in a special way, e.g. coolants or engine oils that must be replenished regularly, the safety instructions of the manufacturer of the hazardous substances or fuels and the applicable regional waste disposal regulations must be observed. Also observe the relevant safety instructions in the product documentation.
6. Depending on the function, certain products such as RF radio equipment can produce an elevated level of electromagnetic radiation. Considering that unborn life requires increased protection, pregnant women should be protected by appropriate measures. Persons with pacemakers may also be endangered by electromagnetic radiation. The employer/operator is required to assess workplaces where there is a special risk of exposure to radiation and, if necessary, take measures to avert the danger.
7. Operating the products requires special training and intense concentration. Make certain that persons who use the products are physically, mentally and emotionally fit enough to handle operating the products; otherwise injuries or material damage may occur. It is the responsibility of the employer to select suitable personnel for operating the products.
8. Prior to switching on the product, it must be ensured that the nominal voltage setting on the product matches the nominal voltage of the AC supply network. If a different voltage is to be set, the power fuse of the product may have to be changed accordingly.
9. In the case of products of safety class I with movable power cord and connector, operation is permitted only on sockets with earthing contact and protective earth connection.
10. Intentionally breaking the protective earth connection either in the feed line or in the product itself is not permitted. Doing so can result in the danger of an electric shock from the product. If extension cords or connector strips are implemented, they must be checked on a regular basis to ensure that they are safe to use.
11. If the product has no power switch for disconnection from the AC supply, the plug of the connecting cable is regarded as the disconnecting device. In such cases, it must be ensured that the power plug is easily reachable and accessible at all times (corresponding to the length of connecting cable, approx. 2 m). Functional or electronic switches are not suitable for providing disconnection from the AC supply. If products without power switches are integrated in racks or systems, a disconnecting device must be provided at the system level.
12. Never use the product if the power cable is damaged. Check the power cable on a regular basis to ensure that it is in proper operating condition. By taking appropriate safety measures and carefully laying the power cable, ensure that the cable cannot be damaged and that no one can be hurt by e.g. tripping over the cable or suffering an electric shock.
13. The product may be operated only from TN/TT supply networks fused with max. 16 A (higher fuse only after consulting with the Rohde & Schwarz group of companies).
14. Do not insert the plug into sockets that are dusty or dirty. Insert the plug firmly and all the way into the socket. Otherwise, this can result in sparks, fire and/or injuries.
15. Do not overload any sockets, extension cords or connector strips; doing so can cause fire or electric shocks.
16. For measurements in circuits with voltages $V_{rms}$ > 30 V, suitable measures (e.g. appropriate measuring equipment, fusing, current limiting, electrical separation, insulation) should be taken to avoid any hazards.
17. Ensure that the connections with information technology equipment comply with IEC 950/EN 60950.
18. Unless expressly permitted, never remove the cover or any part of the housing while the product is in operation. Doing so will expose circuits and components and can lead to injuries, fire or damage to the product.

19. If a product is to be permanently installed, the connection between the PE terminal on site and the product's PE conductor must be made first before any other connection is made. The product may be installed and connected only by a license electrician.
20. For permanently installed equipment without built-in fuses, circuit breakers or similar protective devices, the supply circuit must be fused in such a way that suitable protection is provided for users and products.
21. Do not insert any objects into the openings in the housing that are not designed for this purpose. Never pour any liquids onto or into the housing. This can cause short circuits inside the product and/or electric shocks, fire or injuries.
22. Use suitable overvoltage protection to ensure that no overvoltage (such as that caused by a thunderstorm) can reach the product. Otherwise the operating personnel will be endangered by electric shocks.
23. Rohde & Schwarz products are not protected against penetration of water, unless otherwise specified (see also safety instruction 1.). If this is not taken into account, there exists the danger of electric shock for the user or damage to the product, which can also lead to personal injury.
24. Never use the product under conditions in which condensation has formed or can form in or on the product, e.g. if the product was moved from a cold to a warm environment.
25. Do not close any slots or openings on the product, since they are necessary for ventilation and prevent the product from overheating. Do not place the product on soft surfaces such as sofas or rugs or inside a closed housing, unless this is well ventilated.
26. Do not place the product on heat-generating devices such as radiators or fan heaters. The temperature of the environment must not exceed the maximum temperature specified in the data sheet.
27. Batteries and storage batteries must not be exposed to high temperatures or fire. Keep batteries and storage batteries away from children. Do not short-circuit batteries and storage batteries.
    If batteries or storage batteries are improperly replaced, this can cause an explosion (warning: lithium cells). Replace the battery or storage battery only with the matching Rohde & Schwarz type (see spare parts list). Batteries and storage batteries must be recycled and kept separate from residual waste. Batteries and storage batteries that contain lead, mercury or cadmium are hazardous waste. Observe the national regulations regarding waste disposal and recycling.
28. Please be aware that in the event of a fire, toxic substances (gases, liquids etc.) that may be hazardous to your health may escape from the product.
29. The product can be very heavy. Be careful when moving it to avoid back or other physical injuries.
30. Do not place the product on surfaces, vehicles, cabinets or tables that for reasons of weight or stability are unsuitable for this purpose. Always follow the manufacturer's installation instructions when installing the product and fastening it to objects or structures (e.g. walls and shelves).
31. Handles on the products are designed exclusively for personnel to hold or carry the product. It is therefore not permissible to use handles for fastening the product to or on means of transport such as cranes, fork lifts, wagons, etc. The user is responsible for securely fastening the products to or on the means of transport and for observing the safety regulations of the manufacturer of the means of transport. Noncompliance can result in personal injury or material damage.
32. If you use the product in a vehicle, it is the sole responsibility of the driver to drive the vehicle safely. Adequately secure the product in the vehicle to prevent injuries or other damage in the event of an accident. Never use the product in a moving vehicle if doing so could distract the driver of the vehicle. The driver is always responsible for the safety of the vehicle. The manufacturer assumes no responsibility for accidents or collisions.
33. If a laser product (e.g. a CD/DVD drive) is integrated in a Rohde & Schwarz product, do not use any other settings or functions than those described in the product documentation. Otherwise this may be hazardous to your health, since the laser beam can cause irreversible damage to your eyes. Never try to take such products apart, and never look into the laser beam.

**Por favor lea imprescindiblemente antes de la primera puesta en funcionamiento las siguientes**

# Informaciones de seguridad

El principio del grupo de empresas Rohde & Schwarz consiste en tener nuestros productos siempre al día con los estandards de seguridad y de ofrecer a nuestros clientes el máximo grado de seguridad. Nuestros productos y todos los equipos adicionales son siempre fabricados y examinados según las normas de seguridad vigentes. Nuestra sección de gestión de la seguridad de calidad controla constantemente que sean cumplidas estas normas. El presente producto ha sido fabricado y examinado según el comprobante de conformidad adjunto según las normas de la CE y ha salido de nuestra planta en estado impecable según los estandards técnicos de seguridad. Para poder preservar este estado y garantizar un funcionamiento libre de peligros, el usuario deberá atenerse a todas las informaciones, informaciones de seguridad y notas de alerta. El grupo de empresas Rohde & Schwarz está siempre a su disposición en caso de que tengan preguntas referentes a estas informaciones de seguridad.

Además queda en la responsabilidad del usuario utilizar el producto en la forma debida. Este producto solamente fue elaborado para ser utilizado en la industria y el laboratorio o para fines de campo y de ninguna manera deberá ser utilizado de modo que alguna persona/cosa pueda ser dañada. El uso del producto fuera de sus fines definidos o despreciando las informaciones de seguridad del fabricante queda en la responsabilidad del usuario. El fabricante no se hace en ninguna forma responsable de consecuencias a causa del mal uso del producto.

Se parte del uso correcto del producto para los fines definidos si el producto es utilizado dentro de las instrucciones de la correspondiente documentación de producto y dentro del margen de rendimiento definido (ver hoja de datos, documentación, informaciones de seguridad que siguen). El uso del producto hace necesarios conocimientos profundos y conocimientos parciales del idioma inglés. Por eso se deberá tener en cuenta de exclusivamente autorizar para el uso del producto a personas peritas o debidamente minuciosamente instruidas con los conocimientos citados. Si fuera necesaria indumentaria de seguridad para el uso de productos de R&S, encontrará la información debida en la documentación del producto en el capítulo correspondiente.

## Símbolos y definiciones de seguridad

| | | | | | | | |
|---|---|---|---|---|---|---|---|
| | | | | | | | |
| Ver documen-tación de producto | Informaciones para maquinaria con uns peso de > 18kg | Peligro de golpe de corriente | ¡Advertencia! Superficie caliente | Conexión a conductor protector | Conexión a tierra | Conexión a masa conductora | ¡Cuidado! Elementos de construcción con peligro de carga electroestática |

| | | | | | |
|---|---|---|---|---|---|
| | | | | | |
| potencia EN MARCHA/PARADA | Indicación Stand-by | Corriente continua DC | Corriente alterna AC | Corriente continua/alterna DC/AC | El aparato está protegido en su totalidad por un aislamiento de doble refuerzo |

Tener en cuenta las informaciones de seguridad sirve para tratar de evitar daños y peligros de toda clase. Es necesario de que se lean las siguientes informaciones de seguridad concienzudamente y se tengan en cuenta debidamente antes de la puesta en funcionamiento del producto. También deberán ser tenidas en cuenta las informaciones para la protección de personas que encontrarán en el capítulo correspondiente de la documentación de producto y que también son obligatorias de seguir. En las informaciones de seguridad actuales hemos juntado todos los objetos vendidos por el grupo de empresas Rohde & Schwarz bajo la denominación de „producto“, entre ellos también aparatos, instalaciones así como toda clase de accesorios.

## Palabras de señal y su significado

| | |
|---|---|
| PELIGRO | Identifica un peligro directo con riesgo elevado de provocar muerte o lesiones de gravedad si no se toman las medidas oportunas. |
| ADVERTENCIA | Identifica un posible peligro con riesgo medio de provocar muerte o lesiones (de gravedad) si no se toman las medidas oportunas. |
| ATENCIÓN | Identifica un peligro con riesgo reducido de provocar lesiones de gravedad media o leve si no se toman las medidas oportunas. |
| CUIDADO | Indica la posibilidad de utilizar mal el producto y a consecuencia dañarlo. |
| INFORMACIÓN | Indica una situación en la que deberían seguirse las instrucciones en el uso del producto, pero que no consecuentemente deben de llevar a un daño del mismo. |

Las palabras de señal corresponden a la definición habitual para aplicaciones civiles en el área económica europea. Pueden existir definiciones diferentes a esta definición en otras áreas económicas o en aplicaciones militares. Por eso se deberá tener en cuenta que las palabras de señal aquí descritas sean utilizadas siempre solamente en combinación con la correspondiente documentación de producto y solamente en combinación con el producto correspondiente. La utilización de las palabras de señal en combinación con productos o documentaciones que no les correspondan puede llevar a malinterpretaciones y tener por consecuencia daños en personas u objetos.

## Informaciones de seguridad elementales

1. El producto solamente debe ser utilizado según lo indicado por el fabricante referente a la situación y posición de funcionamiento sin que se obstruya la ventilación. Si no se convino de otra manera, es para los productos R&S válido lo que sigue:
   como posición de funcionamiento se define principialmente la posición con el suelo de la caja para abajo , modo de protección IP 2X, grado de suciedad 2, categoría de sobrecarga eléctrica 2, utilizar solamente en estancias interiores, utilización hasta 2000 m sobre el nivel del mar, transporte hasta 4.500 m sobre el nivel del mar.
   A menos que se especifique otra cosa en la hoja de datos, se aplicará una tolerancia de ±10% sobre el voltaje nominal y de ±5% sobre la frecuencia nominal.
2. En todos los trabajos deberán ser tenidas en cuenta las normas locales de seguridad de trabajo y de prevención de accidentes. El producto solamente debe de ser abierto por personal perito autorizado. Antes de efectuar trabajos en el producto o abrirlo deberá este ser desconectado de la corriente. El ajuste, el cambio de partes, la manutención y la reparación deberán ser solamente efectuadas por electricistas autorizados por R&S. Si se reponen partes con importancia para los aspectos de seguridad (por ejemplo el enchufe, los transformadores o los fusibles), solamente podrán ser sustituidos por partes originales. Despues de cada recambio de partes elementales para la seguridad deberá ser efectuado un control de seguridad (control a primera vista, control de conductor protector, medición de resistencia de aislamiento, medición de medición de la corriente conductora, control de funcionamiento).

3. Como en todo producto de fabricación industrial no puede ser excluido en general de que se produzcan al usarlo elementos que puedan generar alergias, los llamados elementos alergénicos (por ejemplo el níquel). Si se producieran en el trato con productos R&S reacciones alérgicas, como por ejemplo urticaria, estornudos frecuentes, irritación de la conjuntiva o dificultades al respirar, se deberá consultar inmediatamente a un médico para averigurar los motivos de estas reacciones.
4. Si productos / elementos de construcción son tratados fuera del funcionamiento definido de forma mecánica o térmica, pueden generarse elementos peligrosos (polvos de sustancia de metales pesados como por ejemplo plomo, berilio, níquel). La partición elemental del producto, como por ejemplo sucede en el tratamiento de materias residuales, debe de ser efectuada solamente por personal especializado para estos tratamientos. La partición elemental efectuada inadecuadamente puede generar daños para la salud. Se deben tener en cuenta las directivas nacionales referentes al tratamiento de materias residuales.
5. En el caso de que se produjeran agentes de peligro o combustibles en la aplicación del producto que debieran de ser transferidos a un tratamiento de materias residuales, como por ejemplo agentes refrigerantes que deben ser repuestos en periodos definidos, o aceites para motores, deberan ser tenidas en cuenta las prescripciones de seguridad del fabricante de estos agentes de peligro o combustibles y las regulaciones regionales para el tratamiento de materias residuales. Cuiden también de tener en cuenta en caso dado las prescripciones de seguridad especiales en la descripción del producto.
6. Ciertos productos, como por ejemplo las instalaciones de radiación HF, pueden a causa de su función natural, emitir una radiación electromagnética aumentada. En vista a la protección de la vida en desarrollo deberían ser protegidas personas embarazadas debidamente. También las personas con un bypass pueden correr peligro a causa de la radiación electromagnética. El empresario/usario está comprometido a valorar y señalar areas de trabajo en las que se corra un riesgo aumentado de exposición a radiaciones para evitar riesgos.
7. La utilización de los productos requiere instrucciones especiales y una alta concentración en el manejo. Debe de ponerse por seguro de que las personas que manejen los productos estén a la altura de los requerimientos necesarios referente a sus aptitudes físicas, psíquicas y emocionales, ya que de otra manera no se pueden excluir lesiones o daños de objetos. El empresario lleva la responsabilidad de seleccionar el personal usuario apto para el manejo de los productos.
8. Antes de la puesta en marcha del producto se deberá tener por seguro de que la tensión preseleccionada en el producto equivalga a la del la red de distribución. Si es necesario cambiar la preselección de la tensión también se deberán en caso dabo cambiar los fusibles correspondientes del prodcuto.
9. Productos de la clase de seguridad I con alimentación móvil y enchufe individual de producto solamente deberán ser conectados para el funcionamiento a tomas de corriente de contacto de seguridad y con conductor protector conectado.
10. Queda prohibida toda clase de interrupción intencionada del conductor protector, tanto en la toma de corriente como en el mismo producto. Puede tener como consecuencia el peligro de golpe de corriente por el producto. Si se utilizaran cables o enchufes de extensión se deberá poner al seguro, que es controlado su estado técnico de seguridad.
11. Si el producto no está equipado con un interruptor para desconectarlo de la red, se deberá considerar el enchufe del cable de distribución como interruptor. En estos casos deberá asegurar de que el enchufe sea de fácil acceso y nabejo (según la medida del cable de distribución, aproximadamente 2 m). Los interruptores de función o electrónicos no son aptos para el corte de la red eléctrica. Si los productos sin interruptor están integrados en construciones o instalaciones, se deberá instalar el interruptor al nivel de la instalación.

12. No utilice nunca el producto si está dañado el cable eléctrico. Compruebe regularmente el correcto estado de los cables de conexión a red. Asegure a través de las medidas de protección y de instalación adecuadas de que el cable de eléctrico no pueda ser dañado o de que nadie pueda ser dañado por él, por ejemplo al tropezar o por un golpe de corriente.
13. Solamente está permitido el funcionamiento en redes de distribución TN/TT aseguradas con fusibles de como máximo 16 A (utilización de fusibles de mayor amperaje sólo previa consulta con el grupo de empresas Rohde & Schwarz).
14. Nunca conecte el enchufe en tomas de corriente sucias o llenas de polvo. Introduzca el enchufe por completo y fuertemente en la toma de corriente. Si no tiene en consideración estas indicaciones se arriesga a que se originen chispas, fuego y/o heridas.
15. No sobrecargue las tomas de corriente, los cables de extensión o los enchufes de extensión ya que esto pudiera causar fuego o golpes de corriente.
16. En las mediciones en circuitos de corriente con una tensión de entrada de $U_{eff}$ > 30 V se deberá tomar las precauciones debidas para impedir cualquier peligro (por ejemplo medios de medición adecuados, seguros, limitación de tensión, corte protector, aislamiento etc.).
17. En caso de conexión con aparatos de la técnica informática se deberá tener en cuenta que estos cumplan los requisitos de la EC950/EN60950.
18. A menos que esté permitido expresamente, no retire nunca la tapa ni componentes de la carcasa mientras el producto esté en servicio. Esto pone a descubierto los cables y componentes eléctricos y puede causar heridas, fuego o daños en el producto.
19. Si un producto es instalado fijamente en un lugar, se deberá primero conectar el conductor protector fijo con el conductor protector del aparato antes de hacer cualquier otra conexión. La instalación y la conexión deberán ser efecutadas por un electricista especializado.
20. En caso de que los productos que son instalados fijamente en un lugar sean sin protector implementado, autointerruptor o similares objetos de protección, el circuito de suministro de corriente deberá estar protegido de manera que usuarios y productos estén suficientemente protegidos.
21. Por favor, no introduzca ningún objeto que no esté destinado a ello en los orificios de la caja del aparato. No vierta nunca ninguna clase de líquidos sobre o en la caja. Esto puede producir corto circuitos en el producto y/o puede causar golpes de corriente, fuego o heridas.
22. Asegúrese con la protección adecuada de que no pueda originarse en el producto una sobrecarga por ejemplo a causa de una tormenta. Si no se verá el personal que lo utilice expuesto al peligro de un golpe de corriente.
23. Los productos R&S no están protegidos contra el agua si no es que exista otra indicación, ver también punto 1. Si no se tiene en cuenta esto se arriesga el peligro de golpe de corriente para el usario o de daños en el producto lo cual también puede llevar al peligro de personas.
24. No utilice el producto bajo condiciones en las que pueda producirse y se hayan producido líquidos de condensación en o dentro del producto como por ejemplo cuando se desplaza el producto de un lugar frío a un lugar caliente.
25. Por favor no cierre ninguna ranura u orificio del producto, ya que estas son necesarias para la ventilación e impiden que el producto se caliente demasiado. No pongan el producto encima de materiales blandos como por ejemplo sofás o alfombras o dentro de una caja cerrada, si esta no está suficientemente ventilada.
26. No ponga el producto sobre aparatos que produzcan calor, como por ejemplo radiadores o calentadores. La temperatura ambiental no debe superar la temperatura máxima especificada en la hoja de datos.

27. Baterías y acumuladores no deben de ser expuestos a temperaturas altas o al fuego. Guardar baterías y acumuladores fuera del alcance de los niños. No cortocircuitar baterías ni acumuladores. Si las baterías o los acumuladores no son cambiados con la debida atención existirá peligro de explosión (atención celulas de Litio). Cambiar las baterías o los acumuladores solamente por los del tipo R&S correspondiente (ver lista de piezas de recambio). Las baterías y acumuladores deben reutilizarse y no deben acceder a los vertederos. Las baterías y acumuladores que contienen plomo, mercurio o cadmio deben tratarse como residuos especiales. Respete en esta relación las normas nacionales de evacuación y reciclaje.
28. Por favor tengan en cuenta que en caso de un incendio pueden desprenderse del producto agentes venenosos (gases, líquidos etc.) que pueden generar daños a la salud.
29. El producto puede poseer un peso elevado. Muévalo con cuidado para evitar lesiones en la espalda u otras partes corporales.
30. No sitúe el producto encima de superficies, vehículos, estantes o mesas, que por sus características de peso o de estabilidad no sean aptas para él. Siga siempre las instrucciones de instalación del fabricante cuando instale y asegure el producto en objetos o estructuras (por ejemplo paredes y estantes).
31. Las asas instaladas en los productos sirven solamente de ayuda para el manejo que solamente está previsto para personas. Por eso no está permitido utilizar las asas para la sujeción en o sobre medios de transporte como por ejemplo grúas, carretillas elevadoras de horquilla, carros etc. El usuario es responsable de que los productos sean sujetados de forma segura a los medios de transporte y de que las prescripciones de seguridad del fabricante de los medios de transporte sean tenidas en cuenta. En caso de que no se tengan en cuenta pueden causarse daños en personas y objetos.
32. Si llega a utilizar el producto dentro de un vehículo, queda en la responsabilidad absoluta del conductor que conducir el vehículo de manera segura. Asegure el producto dentro del vehículo debidamente para evitar en caso de un accidente las lesiones u otra clase de daños. No utilice nunca el producto dentro de un vehículo en movimiento si esto pudiera distraer al conductor. Siempre queda en la responsabilidad absoluta del conductor la seguridad del vehículo. El fabricante no asumirá ninguna clase de responsabilidad por accidentes o colisiones.
33. Dado el caso de que esté integrado un producto de laser en un producto R&S (por ejemplo CD/DVD-ROM) no utilice otras instalaciones o funciones que las descritas en la documentación de producto. De otra manera pondrá en peligro su salud, ya que el rayo laser puede dañar irreversiblemente sus ojos. Nunca trate de descomponer estos productos. Nunca mire dentro del rayo laser.

## Certified Quality System

# DIN EN ISO 9001 : 2000
# DIN EN 9100 : 2003
# DIN EN ISO 14001 : 2004

DQS REG. NO 001954 QM UM

## QUALITÄTSZERTIFIKAT

*Sehr geehrter Kunde,*
Sie haben sich für den Kauf eines Rohde&Schwarz-Produktes entschieden. Hiermit erhalten Sie ein nach modernsten Fertigungsmethoden hergestelltes Produkt. Es wurde nach den Regeln unseres Managementsystems entwickelt, gefertigt und geprüft.
Das Rohde&Schwarz Managementsystem ist zertifiziert nach:

DIN EN ISO 9001:2000
DIN EN 9100:2003
DIN EN ISO 14001:2004

## CERTIFICATE OF QUALITY

*Dear Customer,*
you have decided to buy a Rohde&Schwarz product. You are thus assured of receiving a product that is manufactured using the most modern methods available. This product was developed, manufactured and tested in compliance with our quality management system standards.
The Rohde&Schwarz quality management system is certified according to:

DIN EN ISO 9001:2000
DIN EN 9100:2003
DIN EN ISO 14001:2004

## CERTIFICAT DE QUALITÉ

*Cher Client,*
vous avez choisi d'acheter un produit Rohde&Schwarz. Vous disposez donc d'un produit fabriqué d'après les méthodes les plus avancées. Le développement, la fabrication et les tests respectent nos normes de gestion qualité.
Le système de gestion qualité de Rohde&Schwarz a été homologué conformément aux normes:

DIN EN ISO 9001:2000
DIN EN 9100:2003
DIN EN ISO 14001:2004

ROHDE&SCHWARZ

1171.0200.11-02.00

# Address List

## Headquarters, Plants and Subsidiaries

### Headquarters

ROHDE&SCHWARZ GmbH & Co. KG
Mühldorfstraße 15 · D-81671 München
P.O.Box 80 14 69 · D-81614 München

Phone +49 (89) 41 29-0
Fax +49 (89) 41 29-121 64
info.rs@rohde-schwarz.com

### Plants

ROHDE&SCHWARZ Messgerätebau GmbH
Riedbachstraße 58 · D-87700 Memmingen
P.O.Box 16 52 · D-87686 Memmingen

Phone +49 (83 31) 1 08-0
+49 (83 31) 1 08-1124
info.rsmb@rohde-schwarz.com

ROHDE&SCHWARZ GmbH & Co. KG
Werk Teisnach
Kaikenrieder Straße 27 · D-94244 Teisnach
P.O.Box 11 49 · D-94240 Teisnach

Phone +49 (99 23) 8 50-0
Fax +49 (99 23) 8 50-174
info.rsdts@rohde-schwarz.com

ROHDE&SCHWARZ závod
Vimperk, s.r.o.
Location Spidrova 49
CZ-38501 Vimperk

Phone +420 (388) 45 21 09
Fax +420 (388) 45 21 13

ROHDE&SCHWARZ GmbH & Co. KG
Dienstleistungszentrum Köln
Graf-Zeppelin-Straße 18 · D-51147 Köln
P.O.Box 98 02 60 · D-51130 Köln

Phone +49 (22 03) 49-0
Fax +49 (22 03) 49 51-229
info.rsdc@rohde-schwarz.com
service.rsdc@rohde-schwarz.com

### Subsidiaries

R&S BICK Mobilfunk GmbH
Fritz-Hahne-Str. 7 · D-31848 Bad Münder
P.O.Box 20 02 · D-31844 Bad Münder

Phone +49 (50 42) 9 98-0
Fax +49 (50 42) 9 98-105
info.bick@rohde-schwarz.com

ROHDE&SCHWARZ FTK GmbH
Wendenschloßstraße 168, Haus 28
D-12557 Berlin

Phone +49 (30) 658 91-122
Fax +49 (30) 655 50-221
info.ftk@rohde-schwarz.com

ROHDE&SCHWARZ SIT GmbH
Am Studio 3
D-12489 Berlin

Phone +49 (30) 658 84-0
Fax +49 (30) 658 84-183
info.sit@rohde-schwarz.com

R&S Systems GmbH
Graf-Zeppelin-Straße 18
D-51147 Köln

Phone +49 (22 03) 49-5 23 25
Fax +49 (22 03) 49-5 23 36
info.rssys@rohde-schwarz.com

GEDIS GmbH
Sophienblatt 100
D-24114 Kiel

Phone +49 (431) 600 51-0
Fax +49 (431) 600 51-11
sales@gedis-online.de

HAMEG Instruments GmbH
Industriestraße 6
D-63533 Mainhausen

Phone +49 (61 82) 800-0
Fax +49 (61 82) 800-100
info@hameg.de

## Locations Worldwide

**Please refer to our homepage: www.rohde-schwarz.com**

- Sales Locations
- Service Locations
- National Websites

# Glossary

## 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 setups.

**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 1 to 4 ports (accessory ZV-Z41). 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 cancelled (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.

# 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.

# 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 fi (i = 1...n) with spacing Delta(f) and the additional condition that f1 = Delta(f). A harmonic grid is required for low pass time domain transforms.

# I

**Isolation error:** Measurement error caused by a crosstalk between the source and receive port of the analyzer.

# L

**Limit check:** Comparison of the measurement results with the limit lines and display of a pass/fail indication. An acoustic warning can be generated in addition if a limit is exceeded.

**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 setup.

**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.

**Power calibration:** Correction method that eliminates frequency response errors in the signal path and in the receiver and ensures accurate power levels and readings; see source power calibration and receiver power calibration.

# R

**Receiver power calibration:** Correction method that the power readings at a specified input port of the analyzer agree with the source power level calibrated at an arbitrary reference plane. Typically, the reference plane is at the input of the receiver so that the calibration eliminates frequency response errors in the calibrated receiver.

**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

**Setup:** A setup comprises a set of diagram areas with all displayed information that can be stored to a NWA setup file (*.zvx). Each setup is displayed in an independent window.

**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).

**Source power calibration:** Correction method that ensures accurate power levels of the generated wave at an arbitrary reference plane in the measurement path. The source power calibration eliminates frequency response errors in the signal path between the source and the reference plane.

**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 NWA to the logical ports used for the measurement of mixed mode S-parameters (balance-unbalance conversion).

**TOSM:** A calibration type using four known standards (through, open, short, match), also called SOLT or 12-term error correction model. TOSM calibration is available for 2, 3 and 4-port measurements.

**Trace:** A trace is a set of data points that can be displayed together on the screen. The trace settings specify the mathematical operations used to obtain traces from the collected data. They complement the definitions of the Channel menu. Each trace is assigned to a channel. The channel settings apply to all traces assigned to the channel.

**Trace point:** Point on the screen which is an element of the displayed trace. The trace points for ratios and wave quantities can be derived from the entire set of measurement points using different detector settings.

**Transmission tracking error:** Frequency-dependent variation of the ratio of the transmitted wave to the reference wave at a test port when an ideal transmission coefficient (= 1) is measured. The transmission tracking error can be corrected by means of a transmission normalization or one of the more sophisticated calibration methods.

**TRL:** A calibration type using a through and a line standard plus a possibly unknown reflect standard. The calibration range is limited. Like TNA and TRM, TRL is especially useful for DUTs in planar line technology.

**TRM:** A calibration type using a through and a match standard plus a possibly unknown reflect standard. Like TRL and TNA, TRM is especially useful for DUTs in planar line technology.

# W

**Window:** Rectangular portion of the screen showing all diagram areas of a particular setup. Windows are limited by a blue frame with several icons. The analyzer uses standard windows provided by the operating system.

# Contents of Chapter 1


**1 Preparing for Use ..... 1.1**

**Front Panel Tour ..... 1.1**

Display ..... 1.2

Disk Drive ..... 1.2

Setup Keys ..... 1.3

Navigation Keys ..... 1.4

Data Entry Keys ..... 1.5

Rotary Knob ..... 1.6

STANDBY Key ..... 1.6

Test Ports ..... 1.7

USB Connector ..... 1.8

Ground Connector ..... 1.8

Front Panel Jumpers ..... 1.9

**Rear Panel Tour ..... 1.10**

**Putting the Instrument into Operation ..... 1.11**

Unpacking the Instrument and Checking the Shipment ..... 1.11

Instrument Setup ..... 1.12

Mounting in a 19" Rack ..... 1.13

EMI Protective Measures ..... 1.13

Connecting the Instrument to the AC Supply ..... 1.13

Power on and off ..... 1.13

Standby and Ready State ..... 1.14

Replacing Fuses ..... 1.14

**Starting the Analyzer, Shutdown ..... 1.15**

**Connecting External Accessories ..... 1.16**

Connecting a Mouse ..... 1.16

Connecting a Keyboard ..... 1.16

Connecting a Printer ..... 1.17

Connecting a Monitor ..... 1.17

Connecting a LAN Cable ..... 1.18

Test Setups with Two LAN Connections ..... 1.19

**Remote Control in a LAN ..... 1.20**

Assigning an IP Address ..... 1.20

Remote Desktop Connection ..... 1.21

**Windows XP ..... 1.22**

**Firmware Update .....1.22**

# 1 Preparing for Use

This chapter gives an overview of the front panel controls and connectors of the ZVA network analyzer and gives all information that is necessary to put the instrument into operation and connect external devices. Notes on reinstallation of the analyzer software appear at the end of the chapter.

**CAUTION!**

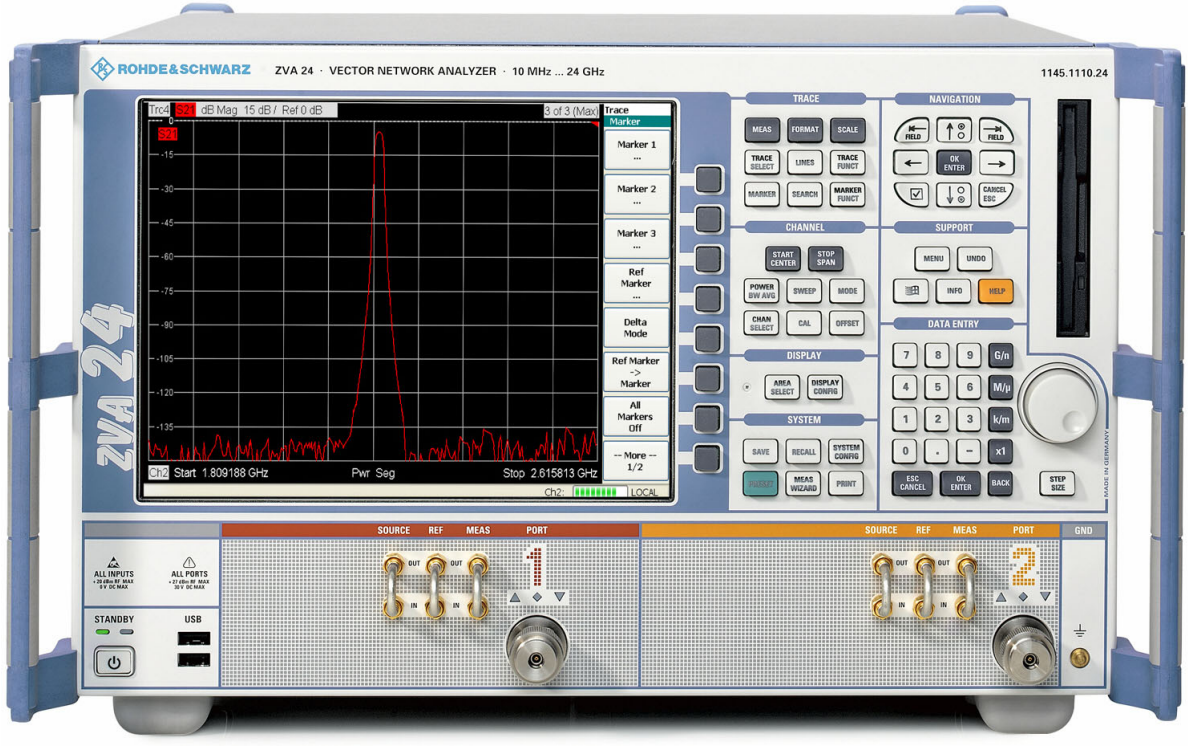
Please make sure to observe the instructions of the following sections so that you cannot cause damage to the instrument or endanger people. This is of particular importance when you use the instrument for the first time. Also observe the general safety instructions at the beginning of this manual.

Chapter 2 of this manual provides an introduction to the operation of the analyzer by means of typical configuration and measurement examples; for a description of the operating concept and an overview of the instrument's capabilities refer to Chapter 3.

For all reference information concerning manual and remote control of the instrument refer to the online help system or its printed/printable version. A more detailed description of the hardware connectors and interfaces is also part of the help system.

## Front Panel Tour

The front panel of the network analyzer consists of the VGA display with the softkey area (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 on the next pages.



## Display

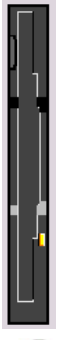
The analyzer is equipped with a color display providing all control elements for the measurements and the diagram areas for the results.

- Refer to section *Navigation Tools of the Screen* in Chapter 3 to learn how to use menus, keys and softkeys.
- Refer to section *Display Elements* in Chapter 3 to obtain information about the results in the diagram area.
- Refer to section *Display Menu* in the online help system and learn how to customize the screen.
- Refer to the data sheet for the technical specifications of the display.

*The screen saver function of the operating system switches off the display if the analyzer receives no command for more than 8 minutes. It is switched on again if any front panel key is pressed. To change the screen saver properties press the Windows key in the SUPPORT keypad and click Control Panel – Display – Screen Saver.*

## Disk Drive

A 3.5" disk drive is located at the front panel of the analyzer.



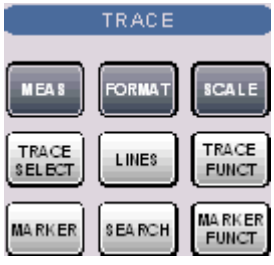
The disk drive can be used for data transfer from and to the analyzer, in particular to do the following:

- Store data files from the analyzer, e.g. store a setup to be used on another instrument or at a later time.
- Load data files, e.g. calibration kits, memory traces, setups.
- Perform a firmware update (if the setup file can not be loaded from another storage medium or a network connection).

***Option Removable Hard Disk, R&S ZVA-B18***
*Option R&S ZVA-B18 replaces the internal hard disk by a removable compact flash card. The option is installed instead of the floppy disk drive. The compact flash card can then be inserted at the front panel of the instrument.*
*If you wish to replace the floppy disk drive of your analyzer by the removable hard disk please contact your R&S service representative. To ensure failure-free operation, avoid placing external cables close to the compact flash card.*

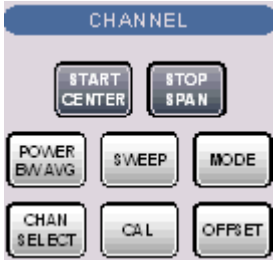
## Setup Keys

The keys in the *TRACE, CHANNEL, DISPLAY, SYSTEM* and *SUPPORT* keypads call up groups of related measurement settings. Each key corresponds to a drop-down menu or menu command of the graphical user interface.



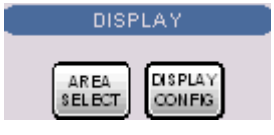

The *TRACE* keys give access to all trace settings and the functions to select, modify and store different traces. In addition the menu provides the marker, search and limit check functions.

- MEAS selects the quantity to be measured and displayed.
- FORMAT defines how the measured data is presented in the graphical display.
- SCALE defines how the current trace is presented in the diagram selected in the Format submenu.
- TRACE SELECT provides functions to handle traces and diagram areas, and assign traces to channels.
- LINES defines limits for measured values and activates the limit check.
- TRACE FUNCT(ions) store traces to the memory and perform mathematical operations on traces.
- MARKER positions markers on a trace, configures their properties and selects the format of the numerical readout.
- SEARCH uses markers to locate specific points on the trace.
- MARKER FUNCT(ions) define the sweep range, scale the diagram and introduce an electrical length offset using the active marker.



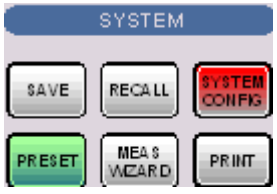

The *CHANNEL* keys give access to all channel settings and the functions to activate, modify and store different channels.

- START CENTER or STOP SPAN define the sweep range, depending on the sweep type.
- POWER BW AVG defines the power of the internal signal source, sets the step attenuators and the IF bandwidths, and configures the sweep average.
- SWEEP defines the scope of measurement, including the sweep type, the trigger conditions and the periodicity of the measurement.
- MODE opens the Port Configuration dialog to define the properties of the physical and logical (balanced) test ports.
- CHAN SELECT provides functions to handle and activate channels.
- CAL provides all functions that are necessary to perform a system error correction (calibration).
- OFFSET provides a selection of length offset parameters to shift the measurement plane.




The *DISPLAY* keys give access to all display settings and to the functions to activate, modify and arrange different diagram areas.

- AREA SELECT provides functions to create and delete diagram areas and select an area as the active area.
- DISPLAY CONFIG provides functions to arrange traces to diagram areas, arrange the diagram areas in the active window and configure the screen and the diagram areas.

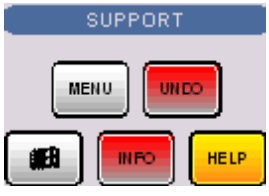



The *SYSTEM* keys give access to the functions to return to a defined instrument state and select general system settings.

- PRESET performs a general factory preset or user preset.
- SYSTEM CONFIG selects general system settings which do not only apply to a particular setup.

  A second group of keys (uncolored) provides standard Windows™ functions to save, recall or print setups and call up the measurement wizard.

- SAVE saves an opened setup to a specific file.
- RECALL recalls an existing setup from a file.
- PRINT prints a setup.




The *SUPPORT* keys give access to the functions to reverse operations, retrieve information on the instrument and obtain assistance.

- UNDO reverses the previous operation.
- INFO calls up a table providing information about the current setup.
- HELP calls up the on-line help system.

  A second group of keys (uncolored) is used to navigate within the graphical user interface:

- MENU sets the cursor to the first item *(File)* in the menu bar of the active application (network analyzer or help system) if no dialog is open. In the network analyzer (NWA) application, menus are equivalent to softkeys and provide fast access to all instrument functions. The menus in the help system are required for accessing all help functions by means of the front panel keys. In NWA dialogs, the MENU key opens the control menu to move or close the dialog.
- The Windows key opens the startup menu of Windows XP from where it is possible to perform system configurations and call up additional software utilities.

## Navigation Keys

The keys in the *NAVIGATION* keypad are used to navigate within the NWA screen and the Help system, to access and control active elements.

The *Left Field* (= Tab) or *Right Field* (= Shift Tab) keys switch between several active elements in dialogs and panes, e.g. in order to access:

- All control elements (e.g. buttons, numerical or text input fields, radio

buttons, checkmarks, combo boxes etc.) in a dialog
- All links in a Help topic

The *Cursor Up* and *Cursor Down* keys are used to:

- Scroll up and down in lists, e.g. among menu items, in a list of keywords, in the Help table of contents, or in the Help topic text
- Increase and decrease numeric input values

*Cursor Up (Down)* become inactive as soon as the beginning of the list is reached. *Cursor Up (Down)* is equivalent to a rotation of the rotary knob to the right (left).

The *Cursor Left* and *Cursor Right* keys are used to:

- Move the cursor to the left or right within input fields
- Compress or expand menus or the Help table of contents
- Move the highlighted item in the menu bar of the active application

OK ENTER is used to:

- Activate the selected active control element, e.g. a button in a dialog or a link in the Help
- Confirm selections and entries made and close dialogs

OK ENTER is equivalent to pressing the rotary knob or the OK ENTER key in the DATA ENTRY keypad.

The *Checkmark* (= Space) key switches a checkmark control in a dialog on or off.

The CANCEL ESC key is used to:

- Close dialogs without activating the entries made (equivalent to the *Close* button)
- Close the Help

CANCEL ESC is equivalent to the CANCEL ESC key in the DATA ENTRY keypad.

## Data Entry Keys

The keys in the DATA ENTRY keypad are used to enter numbers and units.

*The ESC CANCEL and OK ENTER keys are equivalent to the corresponding keys in the NAVIGATION keypad.*
*The remaining 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 change the sign of the entered numeric value. Multiple decimal points are not allowed; pressing – for a second time cancels the effect of the first entry.
- 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 *Data Entry* section in Chapter 3.

- In numeric input fields (e.g. in the numeric entry bar), the *G/n, M/µ, k/m* or *x1* keys multiply the entered value with factors of $10^{(-)9}$, $10^{(-)6}$, $10^{(-)3}$ or 1 and add the appropriate physical unit. *x1* is equivalent to OK ENTER. It confirms the previous entry and deactivates the input field (closes the numeric entry bar).
- In character input fields, the *G/n, M/µ, k/m* keys enter the letters *G, M, K,* respectively. *x1* is equivalent to OK ENTER. It confirms the previous entry and deactivates the input field (closes the numeric entry bar).

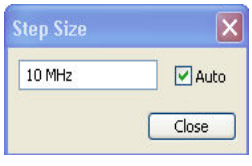
BACK deletes the last character before the cursor position or the selected character sequence. If an entire numeric value is selected, BACK moves the cursor in front of the first digit.

## Rotary Knob

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

NOTE *Turning or pressing the rotary knob is equivalent to the action of the Cursor Up and Cursor Down keys or the ENTER key in the NAVIGATION keypad.*

STEP SIZE opens an input box to select the steps (in units of the current physical parameter) between two consecutive values if the rotary knob is turned to increase or decrease numeric values. See *Dialogs* in Chapter 3.




## 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
- Shut down the instrument.

## Test Ports

N-connectors, numbered 1 to 4 (for four-port instruments). 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.
- With 2, 3 or 4 test ports, it is possible to perform full two-port, 3-port or 4-port measurements; see *S-Parameters* section in Chapter 3.

**Attention!**

The maximum input levels at all test ports according to the front panel labeling or the data sheet must not be exceeded.

In addition, the maximum input voltages of the other input connectors at the front and rear panel must not be exceeded.

The three LEDs above each test port indicate the connector state:

- *The **amber** LED is on while the connector is used as a source port.*
- *The **green** LED is on while the connector is used as a bidirectional (source and receive) port.*
- *The **blue** LED is on while the connector is used as a receive port.*

## USB Connector

Double Universal Serial Bus connector of type A (master USB), used to connect e.g a keyboard, mouse or other pointing devices, the Calibration Unit (accessory R&S ZV-Z51/-Z52 or R&S ZV-Z58), a printer or an external storage device (USB stick, CD-ROM drive etc.).

*The length of passive connecting USB cables should not exceed 1 m. The maximum current per USB port is 500 mA.*

## Ground Connector

Connector providing the ground of the analyzer's supply voltage.

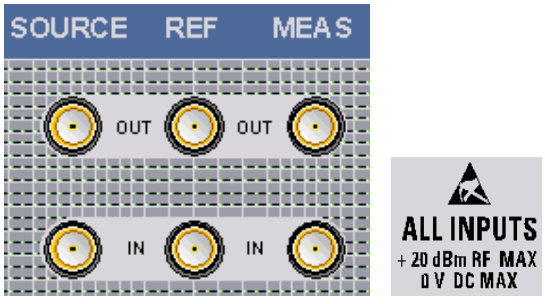
**Attention!**

*Electrostatic discharge (ESD) may cause damage to the electronic components of the DUT and the analyzer. Use the wrist strap and cord supplied with the instrument to connect yourself to the* GND *connector.*

# Front Panel Jumpers

3 pairs of SMA connectors for each test port. The front panel jumpers are available as option R&S ZVA<n>-B16, *Direct Generator/Receiver Access.* For detailed ordering information and application examples refer to the data sheet.

The jumpers give direct access to various RF input and output signals. They can be used to insert external components (e.g. external signal separating devices, power amplifiers, etc.) into the signal path in order to develop custom measurements, e.g. to test high power devices and extend the dynamic range. If no external components are connected, each pair of OUT/IN jumper connectors is through-connected internally.




- The SOURCE OUT signal comes from the internal RF signal source. The SOURCE IN signal goes to the test port. A power amplifier can be inserted between SOURCE OUT and SOURCE IN in order to boost the test port power.
- The REF OUT signal comes from the coupler and provides the reference signal. The REF IN signal goes to the receiver input for the reference signal.
- The MEAS OUT signal comes from the coupler and provides the received (measured) signal. The MEAS IN signal goes to the receiver input for the measured signal.

**Attention!**

The maximum RF input levels at all front panel jumpers inputs according to the front panel labeling or the data sheet must not be exceeded.

In addition, it is important that the signal fed in at the jumper input connectors contains no DC offset, as this may impair the measurements and even cause damage to the instrument.

# Rear Panel Tour

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

The rear connectors are described in detail in the annex *Hardware Interfaces* in the online help system.

- *PORT BIAS* comprises up to four inputs for an external DC voltage (bias) to be applied to the test ports. Each *PORT BIAS* input is protected by an exchangeable fuse.
- *IEC Bus* is the GPIB bus connector (according to standard IEEE 488 / IEC 625).
- *AUX* is an auxiliary connector, to be wired as needed. *AUX* is not fitted on standard instruments.
- *LAN 1* and *LAN 2* are two equivalent connectors to connect the analyzer to a Local Area Network.
- *USB* is a double Universal Serial Bus connector of type A (master USB), used to connect a keyboard, mouse or other pointing device.
- *10 MHz REF* serves as an input or output for the 10 MHz reference clock signal.
- *DC MEAS* comprises two inputs for DC measurements, specified for different voltage ranges.
- *MONITOR* is a sub-Min-D connector used to connect an external VGA monitor.
- *USER CONTROL* is a D-sub connector used as an input and output for low-voltage (3.3 V) TTL control signals.
- *EXT. TRIGGER* is an input for a low-voltage (3.3 V) TTL external trigger signal.

**Attention!**

*The maximum input levels and voltages of the input connectors at the front and rear panel must not be exceeded.*

*The* EXT TRIGGER *input connector and pin 2 of the* USER CONTROL *connector must never be used simultaneously as inputs for external trigger signals.*

# Putting the Instrument into Operation

This section describes the basic steps to be taken when setting up the analyzer for the first time.

**Attention!**

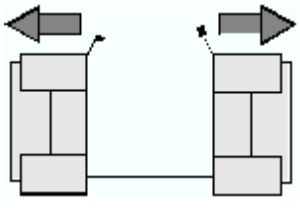
Before turning on the instrument, please make sure that the following conditions are fulfilled:

- Instrument covers are in place and all fasteners are tightened.
- Fan openings are free from obstructions.
- Signal levels at the input connectors are all within the specified maxima.
- Signal outputs are correctly connected and not overloaded.
- The instrument is dry and shows no condensation.

Non-observance may cause damage to the instrument!

## Unpacking the Instrument and Checking the Shipment

1. Unpack the instrument and the other contents of the shipping container.
2. Check the shipment against the list of accessories to ensure that all items are included.
3. Remove the two protective caps from the front and rear and carefully inspect the analyzer to make sure that it was not damaged during shipment.



Should the instrument be damaged, immediately notify the forwarder who shipped the instrument to you and keep the container and packing material.

Equipment returned or sent in for repair must be packed in the original container or packing with electrostatic protection. It is recommended to keep at least the two protective caps for front and rear side in order to prevent damage to the controls and connectors.

# Instrument Setup

The network analyzer is designed for use under laboratory conditions, either on a bench top or in a rack. The general ambient conditions required at the operating site are as follows:

- The ambient temperature must be in the ranges specified for operation and for compliance with specifications (see data sheet).
- All fan openings including the rear panel perforations must be unobstructed. The distance to the wall should be at least 10 cm.

**Attention!**

To avoid damage of electronic components of the DUT and the analyzer, the operating site must be protected against electrostatic discharge (ESD). ESD is most likely to occur when you connect or disconnect a DUT or test fixture to the analyzer's test ports.

To prevent ESD damage use the wrist strap and grounding cord supplied with the instrument and connect yourself to the GND connector at the front panel.

## Bench Top Operation

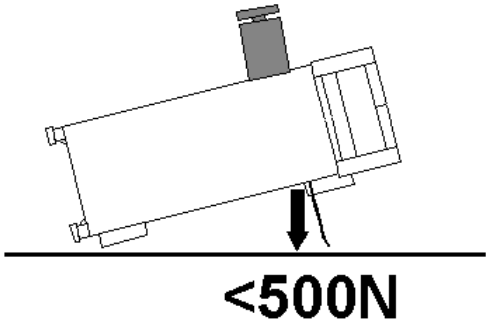
If the analyzer is operated on a bench top, the surface should be flat.

The instrument can be used in horizontal position, standing on its feet, or with the support feet on the bottom extended.

**CAUTION!**

The feet must be fully folded in or out. Only in this way can the stability of analyzer be guaranteed and reliable operation be ensured.

The total weight on the extended feet (including the analyzer's weight and other units placed on top) must not exceed 500 N. The units must be secured against slipping (e.g. by locking the feet at the upper front frame).




When moving the unit with expanded feet, the feet might collapse and fold in. To avoid injuries, the unit must therefore not be moved with the feet out.

## Mounting in a 19" Rack

Using the adapter ZZA-611 (order number 1096.3302.00) the instrument can be mounted in 19" racks according to the mounting instructions supplied with the rack adapter.

**Attention!**

- Allow for sufficient air supply in the rack.
- Make sure that there is sufficient space between the ventilation holes and the rack casing.

## EMI Protective Measures

In order to avoid electromagnetic interference, the instrument may only be operated when it is closed and with all shielding covers fitted. Only appropriate shielded signal and control cables may be used.

## Connecting the Instrument to the AC Supply

The network analyzer is automatically adapted to the AC supply voltage supplied. The supply voltage must be in the range 100 V to 240 V; 50 Hz to 60 Hz. The mains connector is located at the bottom left corner of the rear panel.

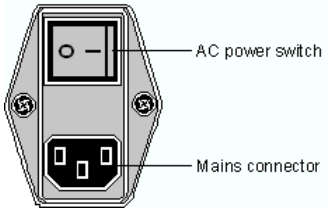
- Connect the network analyzer to the AC power source using the AC power cable delivered with the instrument.

The maximum power consumption of the analyzer is 450 W. The typical power consumption and the frequency range is listed in the data sheet.

The network analyzer is protected by two fuses as specified on the label on the power supply. The fuses are located on an AC Fuse Board (order no. 1145.3906.02) which must be replaced to change the fuses. Replacing the AC Fuse Board requires opening the instrument and is described in the service manual.

## Power on and off

The mains connector is located at the bottom left corner of the rear panel.



- To turn the power on or off, press the AC power switch to position I (On) or 0 (Off).

After power-on, the analyzer is in standby or ready state, depending on the state of the *STANDBY* toggle switch at the front panel of the instrument when the instrument was switched off for the last time.

NOTE *The AC power switch can be permanently on. Switching off is required only if the instrument must be completely removed from the AC power supply.*

## Standby and Ready State

The *STANDBY* toggle switch is located in the bottom left corner of the front panel.

- After switching on the AC power, press the *STANDBY* key briefly to switch the analyzer from the standby to ready state or vice versa.
- In standby state, the right, amber LED is on. The standby power only supplies the power switch circuits and the optional oven quartz (OCXO, 10 MHz reference oscillator, option ZVAB-B4, order no. 1164.1757.02). In this state it is safe to switch off the AC power and disconnect the instrument from the power supply.
- In ready state, the left, green LED is on. The analyzer is ready for operation. All modules are power-supplied and the analyzer initiates its startup procedure.

**Caution!**

The instrument is still power-supplied while it is in standby mode.

## Replacing Fuses

The DC inputs PORT BIAS at the rear panel are each protected by a fuse IEC 127 - F 250 L (250 mA flink).

- To replace the fuses open the fuse holder by slightly turning the lid counterclockwise.

# Starting the Analyzer, Shutdown

To start the analyzer, proceed as follows:

1. Make sure that the instrument is connected to the AC power supply and the power switch on the rear panel is in position I (On).
2. If necessary, press the STANDBY toggle switch on the front panel to switch the instrument to ready state (the green LED is on).

In ready state, the analyzer automatically performs a system check, boots the Windows XP operating system and then starts the network analyzer (NWA) application. If the last analyzer session was terminated regularly, the NWA application uses the last setup with all instrument settings.

To shut down the analyzer, proceed as follows:

1. Press the STANDBY key to save the current setup, close the NWA application, shut down Windows XP and set the instrument to standby state. Of course you can also perform this procedure step by step like in any Windows session.
2. If desired, set the AC power switch to position 0 (Off).

**Attention!**

It is strongly recommended to switch the analyzer to standby state before disconnecting it from the AC supply. If you set the power switch to 0 while the NWA application is still running, you will lose the current settings. Moreover, loss of program data can not be excluded if the application is terminated improperly.

# Connecting External Accessories

The equivalent USB ports on the front and rear panel of the analyzer can be used to connect a variety of accessories:

- A mouse simplifies operation of the instrument using the controls and dialogs of the Graphical User Interface (GUI).
- A keyboard simplifies the entry of data.
- A printer generates hard copies of the screen contents.

In addition the analyzer provides interfaces for monitor connection and network integration:

- An external monitor shows the magnified Graphical User Interface (GUI) with all diagram areas and controls.
- A LAN connection can be established in order to access the hard disk or control the analyzer from an external PC.

## Connecting a Mouse

A USB mouse can be connected to one of the Universal Serial Bus connectors on the front panel or on the rear panel (double USB connectors).

The mouse is detected automatically when it is connected. It is safe to connect or disconnect the mouse during the measurement.

*Use the Start – Control Panel – Mouse menu of Windows XP to configure the mouse properties. To access Windows XP, press the Windows key on the external keyboard or on the front panel.*

*Operating the analyzer does not require a mouse. You can access all essential functions using the keys on the front panel.*

## Connecting a Keyboard

A keyboard can be connected to one of the Universal Serial Bus connectors on the front panel or on the rear panel (double USB connectors).

The keyboard is detected automatically when it is connected. The default input language is English – US. It is safe to connect or disconnect the external keyboard during the measurement.

*Use the Start – Control Panel – Keyboard or Regional and Language Options menu of Windows XP to configure the keyboard properties. To access Windows XP, press the Windows key on the external keyboard or on the front panel.*

*Operating the analyzer does not require a keyboard. You can access all essential functions using the keys on the front panel. In combination with a mouse, the front panel keys provide access to all instrument functions.*

## Connecting a Printer

A printer can be connected to one of the Universal Serial Bus connectors on the front panel or on the rear panel (double USB connectors).

It is safe to connect or disconnect the printer during the measurement. When printing a copy *(File – Print),* the analyzer checks whether a printer is connected and turned on and whether the appropriate printer driver is installed. If required, printer driver installation is initiated using Windows XP's Add Printer Wizard. The wizard is self-explanatory. A printer driver needs to be installed only once, even though the printer may be temporarily removed from the analyzer.

**Printer driver installation**

A great variety of printer drivers is available on the analyzer. To obtain the complete list, access Windows XP (press the *Windows* key) and open the Add Printer Wizard in the *Start – Control Panel – Printer and Faxes* menu.

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 Printer Wizard to complete the installation.

Use the *Page Setup* dialog or the *Start – Control Panel – Printers and Faxes* menu of Windows XP to configure the printer properties. To access Windows XP, press the *Windows* key on the external keyboard or on the front panel.

## Connecting a Monitor

A standard VGA monitor can be connected to the 15-pole Sub-Min-D MONITOR connector on the rear panel of the analyzer.

**Attention!**

The monitor must be connected while the instrument is switched off (in standby mode). Otherwise correct operation can not be guaranteed.

The monitor displays the magnified analyzer screen with all diagram areas, measurement results and control elements. No extra configuration is required.

With an additional mouse or keyboard connected to the analyzer, you can control the measurement from the external monitor. If desired, click *Display – Config/View – Hardkey Bar* to add the Hardkey Bar (front panel key bar) to the analyzer screen.

## Connecting a LAN Cable

A LAN cable can be connected to one of the LAN connectors on the rear panel of the analyzer. To establish a LAN connection proceed as follows:

1. Refer to section *Assigning an IP Address* and learn how to avoid connection errors.
2. Connect an appropriate LAN cable to one of the LAN ports. Use a commercial RJ-45 cable to establish a non-dedicated network connection, or a cross-over RJ-45 cable to establish a dedicated connection between the analyzer and a single PC.

**Dedicated vs. non-dedicated network connections**

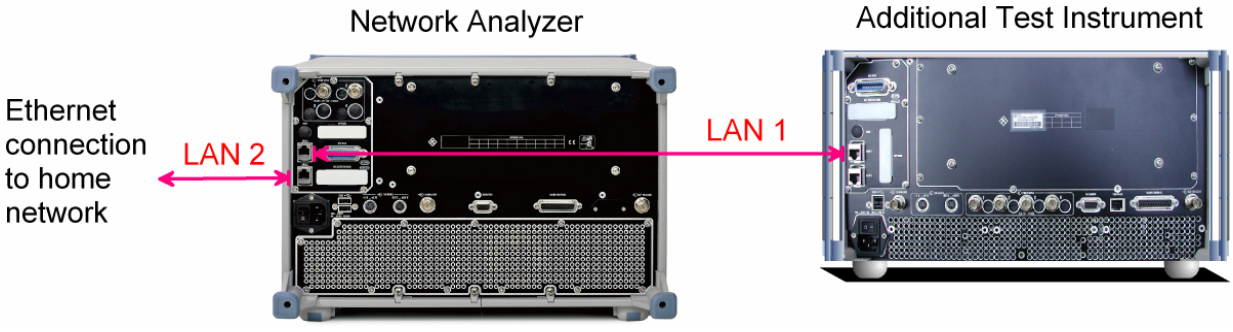
There are two methods to establish a LAN connection of the analyzer:

- A non-dedicated network (Ethernet) connection from the analyzer to an existing network made with an ordinary RJ-45 network cable. The analyzer is assigned an IP address and can coexist with a computer and with other hosts on the same network.
- A dedicated network connection between the analyzer and a single computer made with a cross-over RJ-45 network cable. The computer must be equipped with a network adapter and is directly connected to the analyzer. The use of hubs, switches, or gateways is not needed, however, data transfer is still made using the TCP/IP protocol.

The IP address information is displayed in the *Info – Setup Info* dialog.

# Test Setups with Two LAN Connections

The two LAN connectors on the rear panel of the analyzer are equivalent. With one LAN connector used to establish a connection to a home/company network, the other one can be used to connect an additional instrument, e.g. an additional analyzer or signal generator:




**Defining the network topology: Router vs. network client**

With two LAN connections, it is possible to use the analyzer in two alternative ways:

- As a client participating in two independent networks, one comprising the home network including the analyzer, the second consisting of the additional test instrument plus the analyzer.
- As a data router between the additional test instrument and the home network. This configuration means that the analyzer and the additional test instrument are integrated into a single network.

The network topology is defined in Windows XP's Control Panel – Network Connections – Local Area Connection Status – Local Area Connection Properties – Internet Protocol (TCP/IP) Properties – Advanced TCP/IP Settings dialog. Both instruments must have independent IP addresses; see section Assigning an IP Address below. Contact your LAN administrator for details.

**Attention!**

Never use both LAN connectors to connect the analyzer in parallel to the same network as this will result in connection errors.

# Remote Control in a LAN

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

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

**Attention!**

An efficient virus protection is a prerequisite for secure operation in the network. Never connect your analyzer to an unprotected network because this may cause damage to the instrument software.

To establish the connection proceed as follows:

1. Press the Windows button and open Windows XP's control panel.
2. Select System and open the Remote tab in the System Properties dialog. Enable Allow users to connect remotely to this computer.
3. Assign an IP address to the analyzer following the directions below and connect the analyzer to the network as described in *Connecting a LAN Cable*.
4. Create a *Remote Desktop Connection* using the analyzer's IP address.

*The analyzer uses a user name and password as credentials for remote access. In the factory configuration,* instrument *is preset for both the user name and the password. To protect the analyzer from unauthorized access, it is recommended to change the factory setting.*

## 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 *alternate TCP/IP* configuration, the addresses must be set manually.

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

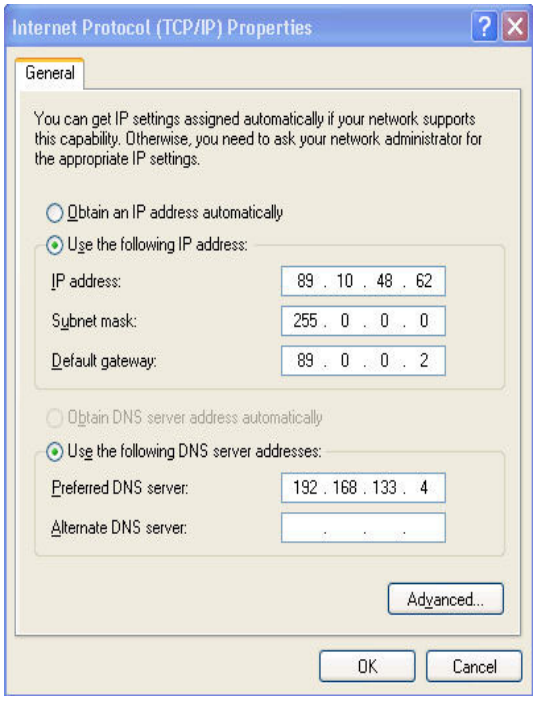
**Attention!**

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

**Manual TCP/IP configuration**

To disable dynamic TCP/IP configuration and enter the TCP/IP address information manually proceed as follows:

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 needed, also obtain the name of your DNS domain and the IP addresses of the DNS and WINS servers on your network. If you use both LAN connectors, you need two different sets of address information.
2. Press the Windows key to access Windows XP.
3. Open the Control Panel – Network Connections – Local Area Connection Status – Local Area Connection Properties – Internet Protocol (TCP/IP) Properties dialog and enter the address information, e.g.:




For more information refer to the Windows XP Help.

## Remote Desktop Connection

*Remote Desktop* is a Windows application which can be used 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 of the applications, files, and network resources of the analyzer.

To set up a *Remote Desktop* connection

1. Connect the analyzer to a LAN and determine its IP address; see *Remote Control in a LAN.*
2. Set up your remote computer (integrated in the LAN) to use *Remote Desktop* and create the *Remote Desktop* connection to the analyzer.

For detailed information about *Remote Desktop* and the connection refer to the Windows XP Help.

# Windows XP

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

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

**Attention!**

The operating system is adapted to the network analyzer. To avoid impairment of instrument functions, only change the settings described in this manual. Existing software must be modified only with update software released by Rohde & Schwarz. Likewise, only programs authorized by Rohde & Schwarz for use on the instrument must be executed.

The *Start* menu of the operating system is accessed by pressing the Windows key in the SUPPORT keypad. All necessary settings can be accessed from the *Start* menu, in particular from the *Control Panel.*

# Firmware Update

Upgrade versions of the analyzer firmware are supplied as single setup files *.msi. To perform a firmware update

1. Copy the setup file to any storage medium accessible from the analyzer. This may be the internal hard disk, an external storage medium (USB memory stick, CD-ROM with external drive) or a network connection (LAN, GPIB bus).
2. Double-click the setup file (or use the front panel keys to select and start the setup file; see Using Front Panel Keys in Chapter 2) and follow the instructions of the setup wizard.

Setup files can be stored and installed again. The default drive name of the internal hard disk is C:. External storage devices are automatically mapped to the next free drive, i.e. D:, E: etc.

**Attention!**

The *Calibration Unit* (accessory R&S ZV-Z51/-Z52 or R&S ZV-Z58) must be disconnected during a firmware update.

# Contents of Chapter 1

1 Preparing for Use ..... 1.1

**Front Panel Tour ..... 1.1**

Display ..... 1.2

Disk Drive ..... 1.2

Setup Keys ..... 1.3

Navigation Keys ..... 1.4

Data Entry Keys ..... 1.5

Rotary Knob ..... 1.6

STANDBY Key ..... 1.7

Test Ports ..... 1.7

USB Connector ..... 1.8

Ground Connector ..... 1.8

**Rear Panel Tour ..... 1.9**

**Putting the Instrument into Operation ..... 1.10**

Unpacking the Instrument and Checking the Shipment ..... 1.10

Instrument Setup ..... 1.10

Mounting in a 19" Rack ..... 1.11

EMI Protective Measures ..... 1.12

Connecting the Instrument to the AC Supply ..... 1.12

Power on and off ..... 1.12

Standby and Ready State ..... 1.13

Replacing Fuses ..... 1.13

**Starting the Analyzer, Shutdown ..... 1.14**

**Connecting External Accessories ..... 1.15**

Connecting a Mouse ..... 1.15

Connecting a Keyboard ..... 1.15

Connecting a Printer ..... 1.16

Connecting a Monitor ..... 1.17

Connecting a LAN Cable ..... 1.17

Test Setups with Two LAN Connections ..... 1.18

**Remote Control in a LAN ..... 1.19**

Assigning an IP Address ..... 1.19

Remote Desktop Connection ..... 1.20

**Windows XP ..... 1.21**

**Firmware Update ..... 1.21**

# 1 Preparing for Use

This chapter gives an overview of the front panel controls and connectors of the ZVB network analyzer and gives all information that is necessary to put the instrument into operation and connect external devices. Notes on reinstallation of the analyzer software appear at the end of the chapter.

**CAUTION!**

Please make sure to observe the instructions of the following sections so that you cannot cause damage to the instrument or endanger people. This is of particular importance when you use the instrument for the first time. Also observe the general safety instructions at the beginning of this manual.

Chapter 2 of this manual provides an introduction to the operation of the analyzer by means of typical configuration and measurement examples; for a description of the operating concept and an overview of the instrument's capabilities refer to Chapter 3.

For all reference information concerning manual and remote control of the instrument refer to the online help system or its printed/printable version. A more detailed description of the hardware connectors and interfaces is also part of the help system.

## Front Panel Tour

The front panel of the network analyzer consists of the VGA display with the softkey area (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 on the next pages.

## Display

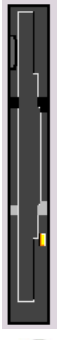
The analyzer is equipped with a color display providing all control elements for the measurements and the diagram areas for the results.

- Refer to section *Navigation Tools of the Screen* in Chapter 3 to learn how to use menus, keys and softkeys.
- Refer to section *Display Elements* in Chapter 3 to obtain information about the results in the diagram area.
- Refer to section *Display Menu* in the online help system and learn how to customize the screen.
- Refer to the data sheet for the technical specifications of the display.

*The screen saver function of the operating system switches off the display if the analyzer receives no command for more than 8 minutes. It is switched on again if any front panel key is pressed. To change the screen saver properties press the Windows key in the SUPPORT keypad and click Control Panel – Display – Screen Saver.*

## Disk Drive

A 3.5" disk drive is located at the front panel of the analyzer.



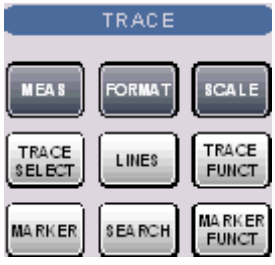
The disk drive can be used for data transfer from and to the analyzer, in particular to do the following:

- Store data files from the analyzer, e.g. store a setup to be used on another instrument or at a later time.
- Load data files, e.g. calibration kits, memory traces, setups.
- Perform a firmware update (if the setup file can not be loaded from another storage medium or a network connection).

***Option Removable Hard Disk, R&S ZVA-B18***
*Option R&S ZVA-B18 replaces the internal hard disk by a removable compact flash card. The option is installed instead of the floppy disk drive. The compact flash card can then be inserted at the front panel of the instrument.*
*If you wish to replace the floppy disk drive of your analyzer by the removable hard disk please contact your R&S service representative. To ensure failure-free operation, avoid placing external cables close to the compact flash card.*

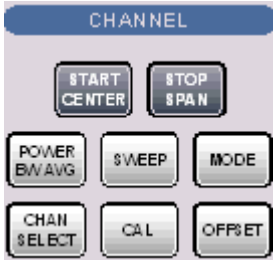
## Setup Keys

The keys in the *TRACE, CHANNEL, DISPLAY, SYSTEM* and *SUPPORT* keypads call up groups of related measurement settings. Each key corresponds to a drop-down menu or menu command of the graphical user interface.



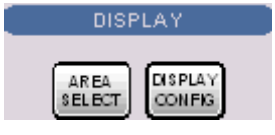

The *TRACE* keys give access to all trace settings and the functions to select, modify and store different traces. In addition the menu provides the marker, search and limit check functions.

- MEAS selects the quantity to be measured and displayed.
- FORMAT defines how the measured data is presented in the graphical display.
- SCALE defines how the current trace is presented in the diagram selected in the Format submenu.
- TRACE SELECT provides functions to handle traces and diagram areas, and assign traces to channels.
- LINES defines limits for measured values and activates the limit check.
- TRACE FUNCT(ions) store traces to the memory and perform mathematical operations on traces.
- MARKER positions markers on a trace, configures their properties and selects the format of the numerical readout.
- SEARCH uses markers to locate specific points on the trace.
- MARKER FUNCT(ions) define the sweep range, scale the diagram and introduce an electrical length offset using the active marker.



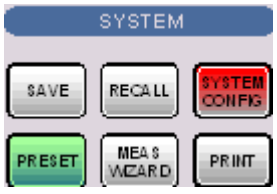

The *CHANNEL* keys give access to all channel settings and the functions to activate, modify and store different channels.

- START CENTER or STOP SPAN define the sweep range, depending on the sweep type.
- POWER BW AVG defines the power of the internal signal source, sets the step attenuators and the IF bandwidths, and configures the sweep average.
- SWEEP defines the scope of measurement, including the sweep type, the trigger conditions and the periodicity of the measurement.
- MODE opens the Port Configuration dialog to define the properties of the physical and logical (balanced) test ports.
- CHAN SELECT provides functions to handle and activate channels.
- CAL provides all functions that are necessary to perform a system error correction (calibration).
- OFFSET provides a selection of length offset parameters to shift the measurement plane.




The *DISPLAY* keys give access to all display settings and to the functions to activate, modify and arrange different diagram areas.

- AREA SELECT provides functions to create and delete diagram areas and select an area as the active area.
- DISPLAY CONFIG provides functions to arrange traces to diagram areas, arrange the diagram areas in the active window and configure the screen and the diagram areas.

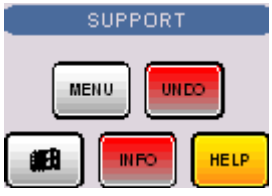



The *SYSTEM* keys give access to the functions to return to a defined instrument state and select general system settings.

- PRESET performs a general factory preset or user preset.
- SYSTEM CONFIG selects general system settings which do not only apply to a particular setup.

  A second group of keys (uncolored) provides standard Windows™ functions to save, recall or print setups and call up the measurement wizard.

- SAVE saves an opened setup to a specific file.
- RECALL recalls an existing setup from a file.
- PRINT prints a setup.




The *SUPPORT* keys give access to the functions to reverse operations, retrieve information on the instrument and obtain assistance.

- UNDO reverses the previous operation.
- INFO calls up a table providing information about the current setup.
- HELP calls up the on-line help system.

  A second group of keys (uncolored) is used to navigate within the graphical user interface:

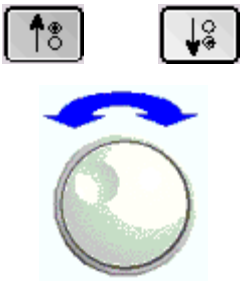
- MENU sets the cursor to the first item *(File)* in the menu bar of the active application (network analyzer or help system) if no dialog is open. In the network analyzer (NWA) application, menus are equivalent to softkeys and provide fast access to all instrument functions. The menus in the help system are required for accessing all help functions by means of the front panel keys. In NWA dialogs, the MENU key opens the control menu to move or close the dialog.
- The Windows key opens the startup menu of Windows XP from where it is possible to perform system configurations and call up additional software utilities.

## Navigation Keys

The keys in the *NAVIGATION* keypad are used to navigate within the NWA screen and the Help system, to access and control active elements.

The *Left Field* (= Tab) or *Right Field* (= Shift Tab) keys switch between several active elements in dialogs and panes, e.g. in order to access:

- All control elements (e.g. buttons, numerical or text input fields, radio buttons, checkmarks, combo boxes etc.) in a dialog
- All links in a Help topic



The *Cursor Up* and *Cursor Down* keys are used to:

- Scroll up and down in lists, e.g. among menu items, in a list of keywords, in the Help table of contents, or in the Help topic text
- Increase and decrease numeric input values

*Cursor Up (Down)* become inactive as soon as the beginning of the list is reached. *Cursor Up (Down)* is equivalent to a rotation of the rotary knob to the right (left).

The *Cursor Left* and *Cursor Right* keys are used to:

- Move the cursor to the left or right within input fields
- Compress or expand menus or the Help table of contents
- Move the highlighted item in the menu bar of the active application

OK ENTER is used to:

- Activate the selected active control element, e.g. a button in a dialog or a link in the Help
- Confirm selections and entries made and close dialogs

OK ENTER is equivalent to pressing the rotary knob or the OK ENTER key in the DATA ENTRY keypad.

The *Checkmark* (= Space) key switches a checkmark control in a dialog on or off.

The CANCEL ESC key is used to:

- Close dialogs without activating the entries made (equivalent to the *Close* button)
- Close the Help

CANCEL ESC is equivalent to the CANCEL ESC key in the DATA ENTRY keypad.

## Data Entry Keys

The keys in the DATA ENTRY keypad are used to enter numbers and units.

*The ESC CANCEL and OK ENTER keys are equivalent to the corresponding keys in the NAVIGATION keypad.*
*The remaining 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 change the sign of the entered numeric value. Multiple decimal points are not allowed; pressing – for a second time cancels the effect of the first entry.
- 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 *Data Entry* section in Chapter 3.

- In numeric input fields (e.g. in the numeric entry bar), the *G/n, M/µ, k/m* or *x1* keys multiply the entered value with factors of $10^{(-)9}$, $10^{(-)6}$, $10^{(-)3}$ or 1 and add the appropriate physical unit. *x1* is equivalent to OK ENTER. It confirms the previous entry and deactivates the input field (closes the numeric entry bar).
- In character input fields, the *G/n, M/µ, k/m* keys enter the letters *G, M, K,* respectively. *x1* is equivalent to OK ENTER. It confirms the previous entry and deactivates the input field (closes the numeric entry bar).

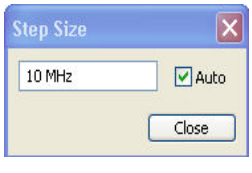
BACK deletes the last character before the cursor position or the selected character sequence. If an entire numeric value is selected, BACK moves the cursor in front of the first digit.

## Rotary Knob

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

*Turning or pressing the rotary knob is equivalent to the action of the Cursor Up and Cursor Down keys or the ENTER key in the NAVIGATION keypad.*

STEP SIZE opens an input box to select the steps (in units of the current physical parameter) between two consecutive values if the rotary knob is turned to increase or decrease numeric values. See *Dialogs* in Chapter 3.



## STANDBY Key

The *STANDBY* toggle switch is located in the bottom left corner of the front panel.

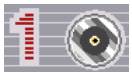
The key serves two main purposes:

- Toggle between standby and ready state
- Shut down the instrument.

## Test Ports

N-connectors, numbered 1 to 4 (for four-port instruments). 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.
- With 2, 3 or 4 test ports, it is possible to perform full two-port, 3-port or 4-port measurements; see *S-Parameters* section in Chapter 3.



**Attention!**

The maximum input levels at all test ports according to the front panel labeling or the data sheet must not be exceeded.

In addition, the maximum input voltages of the other input connectors at the front and rear panel must not be exceeded.

The three LEDs above each test port indicate the connector state:

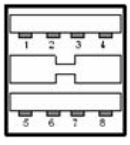
The **amber** LED is on while the connector is used as a source port.

The **green** LED is on while the connector is used as a bidirectional (source and receive) port.

The **blue** LED is on while the connector is used as a receive port.

## USB Connector

Double Universal Serial Bus connector of type A (master USB), used to connect e.g a keyboard, mouse or other pointing devices, the Calibration Unit (accessory R&S ZV-Z51/-Z52 or R&S ZV-Z58), a printer or an external storage device (USB stick, CD-ROM drive etc.).



*The length of passive connecting USB cables should not exceed 1 m. The maximum current per USB port is 500 mA.*

## Ground Connector

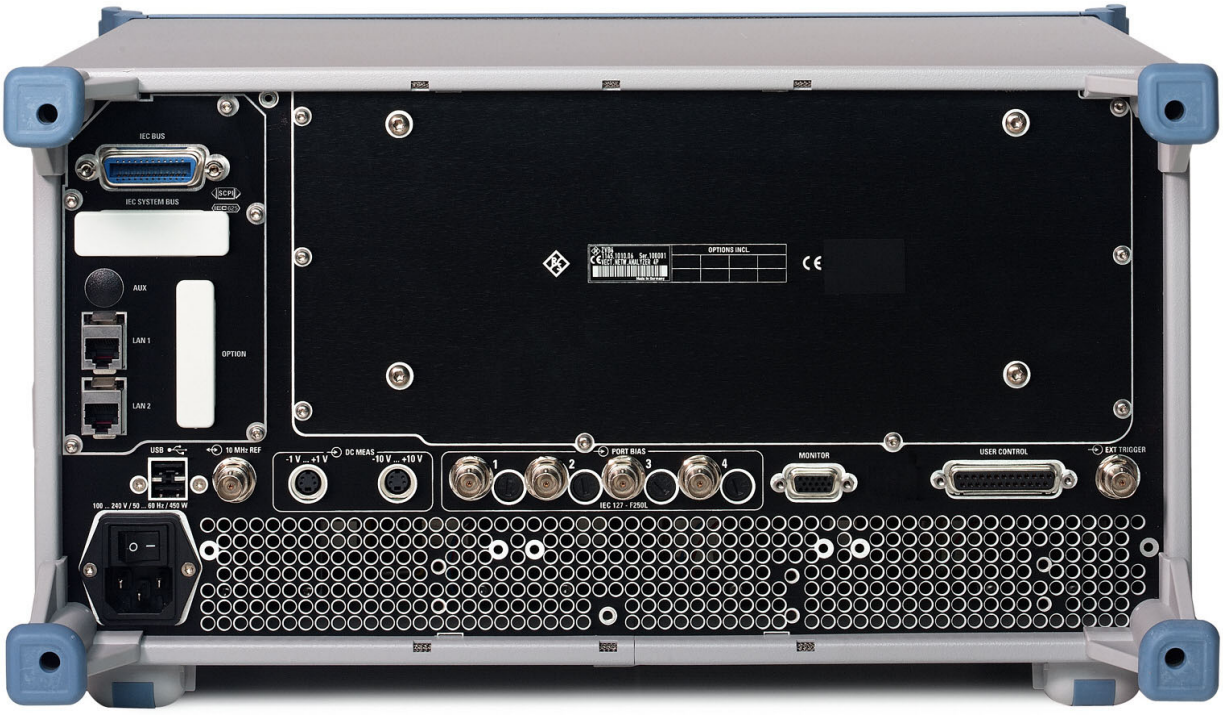
Connector providing the ground of the analyzer's supply voltage.

**Attention!**

*Electrostatic discharge (ESD) may cause damage to the electronic components of the DUT and the analyzer. Use the wrist strap and cord supplied with the instrument to connect yourself to the* GND *connector.*

# Rear Panel Tour

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




The rear connectors are described in detail in the annex *Hardware Interfaces* in the online help system.

- *PORT BIAS* comprises four inputs for an external DC voltage (bias) to be applied to the test ports. Each *PORT BIAS* input is protected by an exchangeable fuse.
- *IEC Bus* is the GPIB bus connector (according to standard IEEE 488 / IEC 625).
- *AUX* is an auxiliary connector, to be wired as needed. *AUX* is not fitted on standard instruments.
- *LAN 1* and *LAN 2* are two equivalent connectors to connect the analyzer to a Local Area Network.
- *USB* is a double Universal Serial Bus connector of type A (master USB), used to connect a keyboard, mouse or other pointing device.
- *10 MHz REF* serves as an input or output for the 10 MHz reference clock signal.
- *DC MEAS* comprises two inputs for DC measurements, specified for different voltage ranges.
- *MONITOR* is a sub-Min-D connector used to connect an external VGA monitor.
- *CASCADE* is a RJ-45 connector for further extensions.
- *USER CONTROL* is a D-sub connector used as an input and output for low-voltage (3.3 V) TTL control signals.
- *EXT. TRIGGER* is an input for a low-voltage (3.3 V) TTL external trigger signal.

**Attention!**

*The maximum input levels and voltages of the input connectors at the front and rear panel must not be exceeded.*

*The* EXT TRIGGER *input connector and pin 2 of the* USER CONTROL *connector must never be used simultaneously as inputs for external trigger signals.*

# Putting the Instrument into Operation

This section describes the basic steps to be taken when setting up the analyzer for the first time.

**Important Note:**

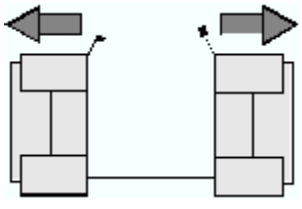
Before turning on the instrument, please make sure that the following conditions are fulfilled:

- Instrument covers are in place and all fasteners are tightened.
- Fan openings are free from obstructions.
- Signal levels at the input connectors are all within the specified maxima.
- Signal outputs are correctly connected and not overloaded.
- The instrument is dry and shows no condensation.

Non-observance may cause damage to the instrument!

## Unpacking the Instrument and Checking the Shipment

1. Unpack the instrument and the other contents of the shipping container.
2. Check the shipment against the list of accessories to ensure that all items are included.
3. Remove the two protective caps from the front and rear and carefully inspect the analyzer to make sure that it was not damaged during shipment.



Should the instrument be damaged, immediately notify the forwarder who shipped the instrument to you and keep the container and packing material.

Equipment returned or sent in for repair must be packed in the original container or packing with electrostatic protection. It is recommended to keep at least the two protective caps for front and rear side in order to prevent damage to the controls and connectors.

## Instrument Setup

The network analyzer is designed for use under laboratory conditions, either on a bench top or in a rack. The general ambient conditions required at the operating site are as follows:

- The ambient temperature must be in the ranges specified for operation and for compliance with specifications (see data sheet).
- All fan openings including the rear panel perforations must be unobstructed. The distance to the wall should be at least 10 cm.

**Attention!**

To avoid damage of electronic components of the DUT and the analyzer, the operating site must be protected against electrostatic discharge (ESD). ESD is most likely to occur when you connect or disconnect a DUT or test fixture to the analyzer's test ports.

To prevent ESD damage use the wrist strap and grounding cord supplied with the instrument and connect yourself to the GND connector at the front panel.

## Bench Top Operation

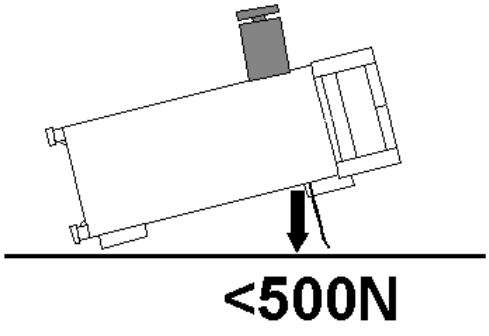
If the analyzer is operated on a bench top, the surface should be flat.

The instrument can be used in horizontal position, standing on its feet, or with the support feet on the bottom extended.

**CAUTION!**

The feet must be fully folded in or out. Only in this way can the stability of analyzer be guaranteed and reliable operation be ensured.

The total weight on the extended feet (including the analyzer's weight and other units placed on top) must not exceed 500 N. The units must be secured against slipping (e.g. by locking the feet at the upper front frame).




When moving the unit with expanded feet, the feet might collapse and fold in. To avoid injuries, the unit must therefore not be moved with the feet out.

# Mounting in a 19" Rack

Using the adapter ZZA-511 (order number 1096.3290.00) the instrument can be mounted in 19" racks according to the mounting instructions supplied with the rack adapter.

**Attention!**

- Allow for sufficient air supply in the rack.
- Make sure that there is sufficient space between the ventilation holes and the rack casing.

## EMI Protective Measures

In order to avoid electromagnetic interference, the instrument may only be operated when it is closed and with all shielding covers fitted. Only appropriate shielded signal and control cables may be used.

## Connecting the Instrument to the AC Supply

The network analyzer is automatically adapted to the AC supply voltage supplied. The supply voltage must be in the range 100 V to 240 V; 50 Hz to 60 Hz. The mains connector is located at the bottom left corner of the rear panel.

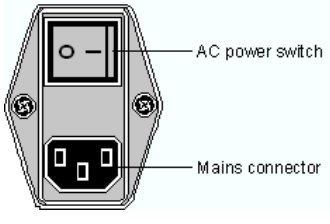
- Connect the network analyzer to the AC power source using the AC power cable delivered with the instrument.

The maximum power consumption of the analyzer is 450 W. The typical power consumption and the frequency range is listed in the data sheet.

The network analyzer is protected by two fuses as specified on the label on the power supply. The fuses are located on an AC Fuse Board (order no. 1145.3906.02) which must be replaced to change the fuses. Replacing the AC Fuse Board requires opening the instrument and is described in the service manual.

## Power on and off

The mains connector is located at the bottom left corner of the rear panel.




- To turn the power on or off, press the AC power switch to position I (On) or 0 (Off).

After power-on, the analyzer is in standby or ready state, depending on the state of the *STANDBY* toggle switch at the front panel of the instrument when the instrument was switched off for the last time.

NOTE

*The AC power switch can be permanently on. Switching off is required only if the instrument must be completely removed from the AC power supply.*

## Standby and Ready State

The *STANDBY* toggle switch is located in the bottom left corner of the front panel.

- After switching on the AC power, press the *STANDBY* key briefly to switch the analyzer from the standby to ready state or vice versa.
- In standby state, the right, amber LED is on. The standby power only supplies the power switch circuits and the optional oven quartz (OCXO, 10 MHz reference oscillator, option ZVAB-B4, order no. 1164.1757.02). In this state it is safe to switch off the AC power and disconnect the instrument from the power supply.
- In ready state, the left, green LED is on. The analyzer is ready for operation. All modules are power-supplied and the analyzer initiates its startup procedure.

**CAUTION!**

The instrument is still power-supplied while it is in standby mode.

## Replacing Fuses

The DC inputs PORT BIAS at the rear panel are protected by four fuses IEC 127 - F 250 L (250 mA flink).

- To replace the fuses open the fuse holder by slightly turning the lid counterclockwise.

Replacement fuses are provided with the instrument.

# Starting the Analyzer, Shutdown

To start the analyzer, proceed as follows:

1. Make sure that the instrument is connected to the AC power supply and the power switch on the rear panel is in position I (On).
2. If necessary, press the STANDBY toggle switch on the front panel to switch the instrument to ready state (the green LED is on).

In ready state, the analyzer automatically performs a system check, boots the Windows XP operating system and then starts the network analyzer (NWA) application. If the last analyzer session was terminated regularly, the NWA application uses the last setup with all instrument settings.

To shut down the analyzer, proceed as follows:

1. Press the STANDBY key to save the current setup, close the NWA application, shut down Windows XP and set the instrument to standby state. Of course you can also perform this procedure step by step like in any Windows session.
2. If desired, set the AC power switch to position 0 (Off).

**Attention!**

It is strongly recommended to switch the analyzer to standby state before disconnecting it from the AC supply. If you set the power switch to 0 while the NWA application is still running, you will lose the current settings. Moreover, loss of program data can not be excluded if the application is terminated improperly.

# Connecting External Accessories

The equivalent USB ports on the front and rear panel of the analyzer can be used to connect a variety of accessories:

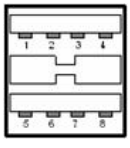
- A mouse simplifies operation of the instrument using the controls and dialogs of the Graphical User Interface (GUI).
- A keyboard simplifies the entry of data.
- A printer generates hard copies of the screen contents.

In addition the analyzer provides interfaces for monitor connection and network integration:

- An external monitor shows the magnified Graphical User Interface (GUI) with all diagram areas and controls.
- A LAN connection can be established in order to access the hard disk or control the analyzer from an external PC.

## Connecting a Mouse

A USB mouse can be connected to one of the Universal Serial Bus connectors on the front panel or on the rear panel (double USB connectors).



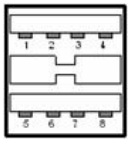
The mouse is detected automatically when it is connected. It is safe to connect or disconnect the mouse during the measurement.

*Use the Start – Control Panel – Mouse menu of Windows XP to configure the mouse properties. To access Windows XP, press the Windows key on the external keyboard or on the front panel.*

*Operating the analyzer does not require a mouse. You can access all essential functions using the keys on the front panel.*

## Connecting a Keyboard

A keyboard can be connected to one of the Universal Serial Bus connectors on the front panel or on the rear panel (double USB connectors).



The keyboard is detected automatically when it is connected. The default input language is English – US. It is safe to connect or disconnect the external keyboard during the measurement.

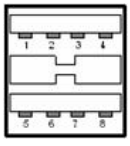
*Use the Start – Control Panel – Keyboard or Regional and Language Options menu of Windows*

*XP to configure the keyboard properties. To access Windows XP, press the Windows key on the external keyboard or on the front panel.*

*Operating the analyzer does not require a keyboard. You can access all essential functions using the keys on the front panel. In combination with a mouse, the front panel keys provide access to all instrument functions.*

## Connecting a Printer

A printer can be connected to one of the Universal Serial Bus connectors on the front panel or on the rear panel (double USB connectors).



It is safe to connect or disconnect the printer during the measurement. When printing a copy *(File – Print),* the analyzer checks whether a printer is connected and turned on and whether the appropriate printer driver is installed. If required, printer driver installation is initiated using Windows XP's Add Printer Wizard. The wizard is self-explanatory. A printer driver needs to be installed only once, even though the printer may be temporarily removed from the analyzer.

**Printer driver installation**

A great variety of printer drivers is available on the analyzer. To obtain the complete list, access Windows XP (press the *Windows* key) and open the Add Printer Wizard in the *Start – Control Panel – Printer and Faxes* menu.

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 Printer Wizard to complete the installation.

Use the *Page Setup* dialog or the *Start – Control Panel – Printers and Faxes* menu of Windows XP to configure the printer properties. To access Windows XP, press the *Windows* key on the external keyboard or on the front panel.

## Connecting a Monitor

A standard VGA monitor can be connected to the 15-pole Sub-Min-D MONITOR connector on the rear panel of the analyzer.

**Attention!**

The monitor must be connected while the instrument is switched off (in standby mode). Otherwise correct operation can not be guaranteed.

The monitor displays the magnified analyzer screen with all diagram areas, measurement results and control elements. No extra configuration is required.

With an additional mouse or keyboard connected to the analyzer, you can control the measurement from the external monitor. If desired, click *Display – Config/View – Hardkey Bar* to add the Hardkey Bar (front panel key bar) to the analyzer screen.

## Connecting a LAN Cable

A LAN cable can be connected to one of the LAN connectors on the rear panel of the analyzer. To establish a LAN connection proceed as follows:

1. Refer to section *Assigning an IP Address* and learn how to avoid connection errors.
2. Connect an appropriate LAN cable to one of the LAN ports. Use a commercial RJ-45 cable to establish a non-dedicated network connection, or a cross-over RJ-45 cable to establish a dedicated connection between the analyzer and a single PC.

**Dedicated vs. non-dedicated network connections**

There are two methods to establish a LAN connection of the analyzer:

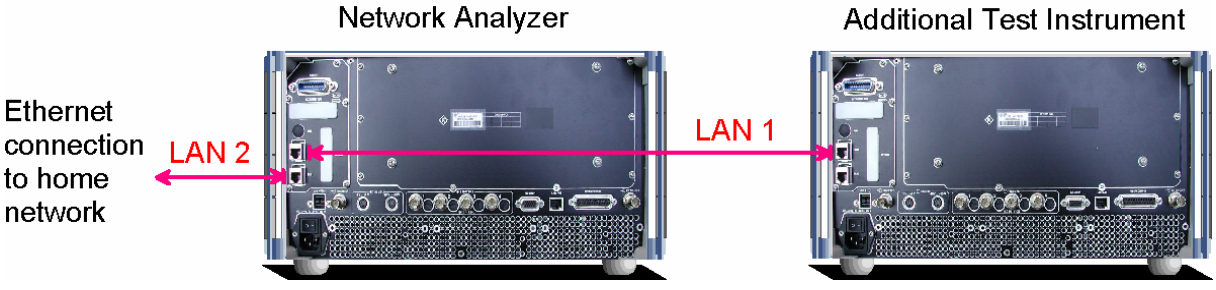
- A non-dedicated network (Ethernet) connection from the analyzer to an existing network made with an ordinary RJ-45 network cable. The analyzer is assigned an IP address and can coexist with a computer and with other hosts on the same network.
- A dedicated network connection between the analyzer and a single computer made with a cross-over RJ-45 network cable. The computer must be equipped with a network adapter and is directly connected to the analyzer. The use of hubs, switches, or gateways is not needed, however, data

transfer is still made using the TCP/IP protocol.

The IP address information is displayed in the *Info – Setup Info* dialog.

## Test Setups with Two LAN Connections

The two LAN connectors on the rear panel of the analyzer are equivalent. With one LAN connector used to establish a connection to a home/company network, the other one can be used to connect an additional instrument, e.g. an additional analyzer or signal generator:




**Defining the network topology: Router vs. network client**

With two LAN connections, it is possible to use the analyzer in two alternative ways:

- As a client participating in two independent networks, one comprising the home network including the analyzer, the second consisting of the additional test instrument plus the analyzer.
- As a data router between the additional test instrument and the home network. This configuration means that the analyzer and the additional test instrument are integrated into a single network.

The network topology is defined in Windows XP's Control Panel – Network Connections – Local Area Connection Status – Local Area Connection Properties – Internet Protocol (TCP/IP) Properties – Advanced TCP/IP Settings dialog. Both instruments must have independent IP addresses; see section Assigning an IP Address below. Contact your LAN administrator for details.

**Attention!**

Never use both LAN connectors to connect the analyzer in parallel to the same network as this will result in connection errors.

# Remote Control in a LAN

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

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

**Attention!**

An efficient virus protection is a prerequisite for secure operation in the network. Never connect your analyzer to an unprotected network because this may cause damage to the instrument software.

To establish the connection proceed as follows:

1. Press the Windows button and open Windows XP's control panel.
2. Select System and open the Remote tab in the System Properties dialog. Enable Allow users to connect remotely to this computer.
3. Assign an IP address to the analyzer following the directions below and connect the analyzer to the network as described in *Connecting a LAN Cable*.
4. Create a *Remote Desktop Connection* using the analyzer's IP address.

*The analyzer uses a user name and password as credentials for remote access. In the factory configuration,* instrument *is preset for both the user name and the password. To protect the analyzer from unauthorized access, it is recommended to change the factory setting.*

## 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 *alternate TCP/IP* configuration, the addresses must be set manually.

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

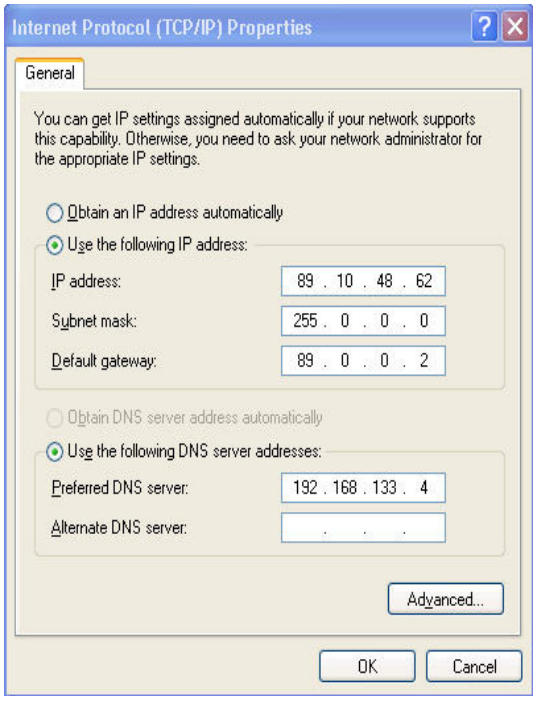
**Attention!**

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

**Manual TCP/IP configuration**

To disable dynamic TCP/IP configuration and enter the TCP/IP address information manually proceed as follows:

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 needed, also obtain the name of your DNS domain and the IP addresses of the DNS and WINS servers on your network. If you use both LAN connectors, you need two different sets of address information.
2. Press the Windows key to access Windows XP.
3. Open the Control Panel – Network Connections – Local Area Connection Status – Local Area Connection Properties – Internet Protocol (TCP/IP) Properties dialog and enter the address information, e.g.:




For more information refer to the Windows XP Help.

## Remote Desktop Connection

*Remote Desktop* is a Windows application which can be used 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 of the applications, files, and network resources of the analyzer.

To set up a *Remote Desktop* connection

1. Connect the analyzer to a LAN and determine its IP address; see *Remote Control in a LAN.*
2. Set up your remote computer (integrated in the LAN) to use *Remote Desktop* and create the *Remote Desktop* connection to the analyzer.

For detailed information about *Remote Desktop* and the connection refer to the Windows XP Help.

# Windows XP

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

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

**Attention!**

The operating system is adapted to the network analyzer. To avoid impairment of instrument functions, only change the settings described in this manual. Existing software must be modified only with update software released by Rohde & Schwarz. Likewise, only programs authorized by Rohde & Schwarz for use on the instrument must be executed.

The *Start* menu of the operating system is accessed by pressing the Windows key in the SUPPORT keypad. All necessary settings can be accessed from the *Start* menu, in particular from the *Control Panel.*

# Firmware Update

Upgrade versions of the analyzer firmware are supplied as single setup files *.msi. To perform a firmware update

1. Copy the setup file to any storage medium accessible from the analyzer. This may be the internal hard disk, an external storage medium (USB memory stick, CD-ROM with external drive) or a network connection (LAN, GPIB bus).
2. Double-click the setup file (or use the front panel keys to select and start the setup file; see Using Front Panel Keys in Chapter 2) and follow the instructions of the setup wizard.

Setup files can be stored and installed again. The default drive name of the internal hard disk is C:. External storage devices are automatically mapped to the next free drive, i.e. D:, E: etc.

**Attention!**

The *Calibration Unit* (accessory R&S ZV-Z51/-Z52 or R&S ZV-Z58) must be disconnected during a firmware update.

# Contents of Chapter 1


**1 Preparing for Use ..... 1.1**

**Front Panel Tour ..... 1.1**

STANDBY Key ..... 1.2

USB Connector ..... 1.2

Ground Connector ..... 1.3

Test Ports ..... 1.3

Front Panel Jumpers ..... 1.4

**Rear Panel Tour ..... 1.5**

**Putting the Instrument into Operation ..... 1.6**

Unpacking the Instrument and Checking the Shipment ..... 1.6

Instrument Setup ..... 1.7

EMI Protective Measures ..... 1.7

Connecting the Instrument to the AC Supply ..... 1.8

Power on and off ..... 1.8

Standby and Ready State ..... 1.8

Replacing Fuses ..... 1.9

**Starting the Analyzer, Shutdown ..... 1.10**

**Connecting External Accessories ..... 1.10**

Connecting a Mouse ..... 1.11

Connecting a Keyboard ..... 1.11

Connecting a Printer ..... 1.11

Connecting a Monitor ..... 1.12

Connecting a LAN Cable ..... 1.13

Test Setups with Two LAN Connections ..... 1.13

**Remote Control in a LAN ..... 1.14**

Assigning an IP Address ..... 1.15

Remote Desktop Connection ..... 1.16

**Windows XP ..... 1.16**

**Firmware Update ..... 1.17**

# 1 Preparing for Use

This chapter gives an overview of the front panel controls and connectors of the network analyzer and gives all information that is necessary to put the instrument into operation and connect external devices. Notes on reinstallation of the analyzer software appear at the end of the chapter.

The ZVT is primarily intended to be remote-controlled via the GPIB interface or a LAN connection. Alternatively it is possible to control the instrument manually using an external monitor in combination with a mouse (see section *Connecting External Accessories* on p. 1.10) or a LAN connection (see section *Remote Control in a LAN* on p. 1.14).

**Attention!**

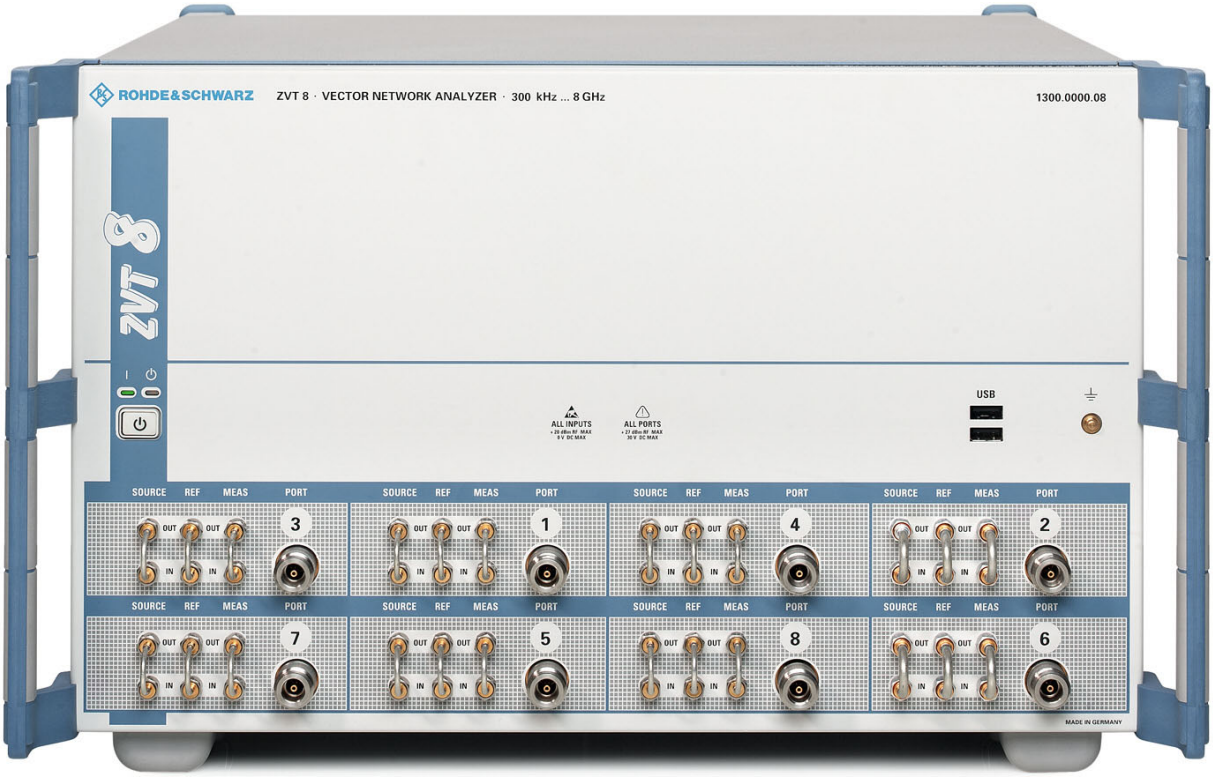
Please make sure to observe the instructions of the following sections so that you cannot cause damage to the instrument or endanger people. This is of particular importance when you use the instrument for the first time. Also observe the general safety instructions at the beginning of this manual.

## Front Panel Tour

The front panel of the R&S ZVT network analyzer contains the following controls and connectors:

- The STANDBY toggle switch used to initiate the startup procedure or switch the analyzer to standby mode.
- A double USB connector to connect pointing or mass storage devices.
- The ground (GND) connector providing the ground of the analyzer's supply voltage.
- Tow RF test ports (PORT 1, PORT 2) serving as outputs for the RF stimulus signals and as inputs for the measured RF signals (response signals). Each test port is complemented by three pairs of front panel jumpers used to test high power devices and extend the dynamic range. Up to 6 additional test ports with jumpers can be purchased as options (order no. 1300.1506.03 to 1300.1506.08; see detailed ordering information in the data sheet).

Brief explanations on the controls and connectors and on the rear panel can be found on the next pages.




## STANDBY Key

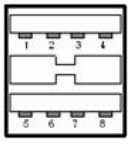
The *STANDBY* toggle switch is located near the left edge of the front panel.

The key serves two main purposes:

- Toggle between standby and ready state
- Shut down the instrument.

## USB Connector

Double Universal Serial Bus connector of type A (master USB), used to connect e.g. a keyboard, mouse or other pointing devices, the Calibration Unit (accessory R&S ZV-Z51/-Z52 or R&S ZV-Z58), a printer or an external storage device (USB stick, CD-ROM drive etc.).



*The length of passive connecting USB cables should not exceed 1 m. The maximum current per USB port is 500 mA.*

## Ground Connector

Connector providing the ground of the analyzer's supply voltage.

**Attention!**

*Electrostatic discharge (ESD) may cause damage to the electronic components of the DUT and the analyzer. Use the wrist strap and cord supplied with the instrument to connect yourself to the* GND *connector.*

## Test Ports

N-connectors, numbered 1 and 2 for two-port instruments. The test ports serve as outputs for the RF stimulus signal and as inputs for the measured RF signals from the DUT (response signals). Up to 6 additional test ports can be purchased as options (order no. 1300.1506.03 to 1300.1506.08; see detailed ordering information in the data sheet).

- With a single test port, it is possible to generate a stimulus signal and measure the response signal in reflection.
- With 2 or more test ports, it is possible to perform full two-port or multiport measurements.
- Each test port is complemented by three pairs of front panel jumpers used to test high power devices and extend the dynamic range, see section *Front Panel Jumpers below*.

**Attention!**

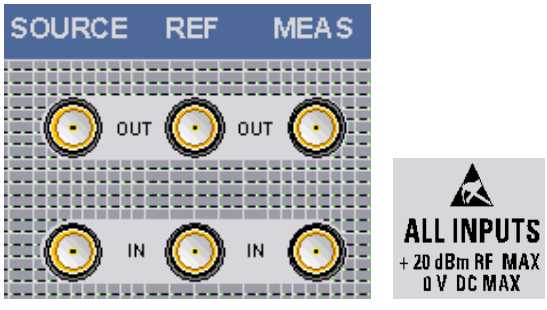
The maximum input levels at all test ports according to the front panel labeling or the data sheet must not be exceeded.

In addition, the maximum input voltages of the other input connectors at the front and rear panel must not be exceeded.

## Front Panel Jumpers

3 pairs of SMA connectors for each test port. The front panel jumpers are available as option R&S ZVA<n>-B16, *Direct Generator/Receiver Access.* For detailed ordering information and application examples refer to the data sheet.

The jumpers give direct access to various RF input and output signals. They can be used to insert external components (e.g. external signal separating devices, power amplifiers, etc.) into the signal path in order to develop custom measurements, e.g. to test high power devices and extend the dynamic range. If no external components are connected, each pair of OUT/IN jumper connectors is through-connected internally.




- The SOURCE OUT signal comes from the internal RF signal source. The SOURCE IN signal goes to the test port. A power amplifier can be inserted between SOURCE OUT and SOURCE IN in order to boost the test port power.
- The REF OUT signal comes from the coupler and provides the reference signal. The REF IN signal goes to the receiver input for the reference signal.
- The MEAS OUT signal comes from the coupler and provides the received (measured) signal. The MEAS IN signal goes to the receiver input for the measured signal.

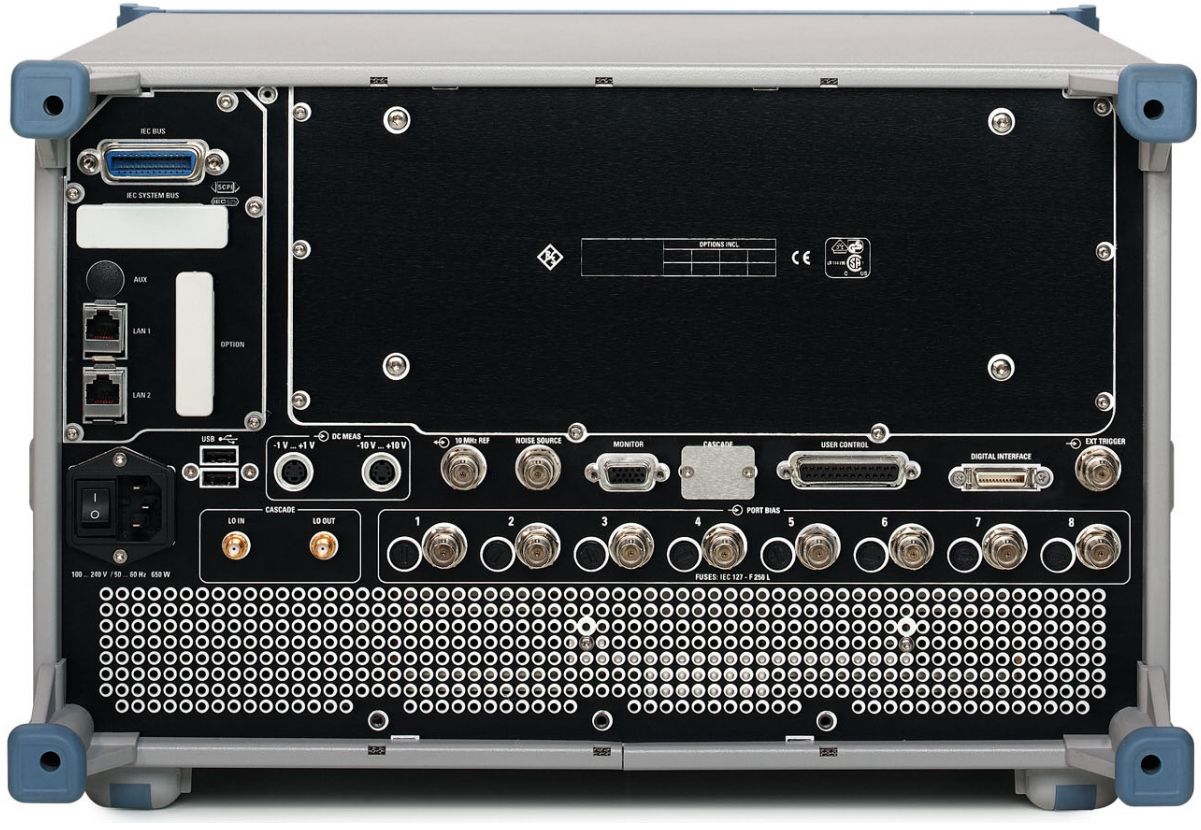
**Attention!**

The maximum RF input levels at all front panel jumpers inputs according to the front panel labeling or the data sheet must not be exceeded.

In addition, it is important that the signal fed in at the jumper input connectors contains no DC offset, as this may impair the measurements and even cause damage to the instrument.

# Rear Panel Tour

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




The rear connectors are described in detail in the annex *Hardware Interfaces* in the online help system.

- The *PORT BIAS* panel contains inputs for an external DC voltage (bias) to be applied to the test ports. A separate input is provided for each test port. Each *PORT BIAS* input is protected by an exchangeable fuse.
- *IEC Bus* is the GPIB bus connector (according to standard IEEE 488 / IEC 625).
- *LAN 1* and *LAN 2* are two equivalent connectors to connect the analyzer to a Local Area Network.
- *USB* is a double Universal Serial Bus connector of type A (master USB), used to connect a keyboard, mouse or other pointing device.
- *DC MEAS* comprises two inputs for DC measurements, specified for different voltage ranges.
- *10 MHz REF* serves as an input or output for the 10 MHz reference clock signal.
- *MONITOR* is a sub-Min-D connector used to connect an external VGA monitor.
- *USER CONTROL* is a D-sub connector used as an input and output for low-voltage (3.3 V) TTL control signals.
- *EXT. TRIGGER* is an input for a low-voltage (3.3 V) TTL external trigger signal.
- The *Noise Source*, *Digital Interface*, and *Cascade* connectors are for future extensions.

The remaining connectors are intended for future extensions and not fitted on standard instruments.

**Attention!**

The maximum input levels and voltages of the input connectors at the front and rear panel must not be exceeded.

The *EXT TRIGGER* input connector and pin 2 of the *USER CONTROL* connector must never be used simultaneously as inputs for external trigger signals.

# Putting the Instrument into Operation

This section describes the basic steps to be taken when setting up the analyzer for the first time.

**Important Note:**

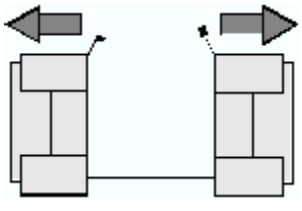
Before turning on the instrument, please make sure that the following conditions are fulfilled:

- Instrument covers are in place and all fasteners are tightened.
- Fan openings are free from obstructions.
- Signal levels at the input connectors are all within the specified maxima.
- Signal outputs are correctly connected and not overloaded.
- The instrument is dry and shows no condensation.

Non-observance may cause damage to the instrument!

## Unpacking the Instrument and Checking the Shipment

1. Unpack the instrument and the other contents of the shipping container.
2. Check the shipment against the list of accessories to ensure that all items are included.
3. Remove the two protective caps from the front and rear and carefully inspect the analyzer to make sure that it was not damaged during shipment.



Should the instrument be damaged, immediately notify the forwarder who shipped the instrument to you and keep the container and packing material.

Equipment returned or sent in for repair must be packed in the original container or packing with electrostatic protection. It is recommended to keep at least the two protective caps for front and rear side in order to prevent damage to the controls and connectors.

# Instrument Setup

The network analyzer is designed for use under laboratory conditions, either on a bench top or in a rack. The general ambient conditions required at the operating site are as follows:

- The ambient temperature must be in the ranges specified for operation and for compliance with specifications (see data sheet).
- All fan openings including the rear panel perforations must be unobstructed. The distance to the wall should be at least 10 cm.

**Attention!**

To avoid damage of electronic components of the DUT and the analyzer, the operating site must be protected against electrostatic discharge (ESD). ESD is most likely to occur when you connect or disconnect a DUT or test fixture to the analyzer's test ports.

To prevent ESD damage use the wrist strap and grounding cord supplied with the instrument and connect yourself to the GND connector at the front panel.

## Bench Top Operation

If the analyzer is operated on a bench top, the surface should be flat. The instrument can be used in horizontal position, standing on its feet.

## Mounting in a 19" Rack

Using the adapter ZZA-611 (order number 1096.3302.00) the instrument can be mounted in 19" racks according to the mounting instructions supplied with the rack adapter.

**Attention!**

- Allow for sufficient air supply in the rack.
- Make sure that there is sufficient space between the ventilation holes and the rack casing.

# EMI Protective Measures

In order to avoid electromagnetic interference, the instrument may only be operated when it is closed and with all shielding covers fitted. Only appropriate shielded signal and control cables may be used.

## Connecting the Instrument to the AC Supply

The network analyzer is automatically adapted to the AC supply voltage supplied. The supply voltage must be in the range 100 V to 240 V; 50 Hz to 60 Hz. The mains connector is located at the bottom left corner of the rear panel.

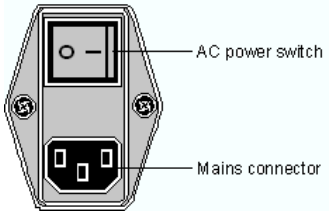
- Connect the network analyzer to the AC power source using the AC power cable delivered with the instrument.

The maximum power consumption of the analyzer is 650 W. The typical power consumption and the frequency range is listed in the data sheet.

The network analyzer is protected by two fuses as specified on the label on the power supply. The fuses are located on an AC Fuse Board (order no. 1145.3906.02) which must be replaced to change the fuses. Replacing the AC Fuse Board requires opening the instrument and is described in the service manual.

## Power on and off

The mains connector is located at the bottom left corner of the rear panel.




- To turn the power on or off, press the AC power switch to position I (On) or 0 (Off).

After power-on, the analyzer is in standby or ready state, depending on the state of the *STANDBY* toggle switch at the front panel of the instrument when the instrument was switched off for the last time.

NOTE *The AC power switch can be permanently on. Switching off is required only if the instrument must be completely removed from the AC power supply.*

## Standby and Ready State

The *STANDBY* toggle switch is located in the bottom left corner of the front panel.

- After switching on the AC power, press the *STANDBY* key briefly to switch the analyzer from the standby to ready state or vice versa.

- In standby state, the right, amber LED is on. The standby power only supplies the power switch circuits and the optional oven quartz (OCXO, 10 MHz reference oscillator, option ZVAB-B4, order no. 1164.1757.02). In this state it is safe to switch off the AC power and disconnect the instrument from the power supply.

- In ready state, the left, green LED is on. The analyzer is ready for operation. All modules are power-supplied and the analyzer initiates its startup procedure.

**CAUTION!**

The instrument is still power-supplied while it is in standby mode.

## Replacing Fuses

The DC inputs PORT BIAS at the rear panel are protected by four fuses IEC 127 - F 250 L (250 mA flink).

- To replace the fuses open the fuse holder by slightly turning the lid counterclockwise.

Replacement fuses are provided with the instrument.

# Starting the Analyzer, Shutdown

To start the analyzer, proceed as follows:

1. Make sure that the instrument is connected to the AC power supply and the power switch on the rear panel is in position I (On).
2. If necessary, press the STANDBY toggle switch on the front panel to switch the instrument to ready state (the green LED is on).

In ready state, the analyzer automatically performs a system check, boots the Windows XP operating system and then starts the network analyzer (NWA) application. If the last analyzer session was terminated regularly, the NWA application uses the last setup with all instrument settings.

To shut down the analyzer, proceed as follows:

1. Press the STANDBY key to save the current setup, close the NWA application, shut down Windows XP and set the instrument to standby state. Of course you can also perform this procedure step by step like in any Windows session.
2. If desired, set the AC power switch to position 0 (Off).

**Attention!**

It is strongly recommended to switch the analyzer to standby state before disconnecting it from the AC supply. If you set the power switch to 0 while the NWA application is still running, you will lose the current settings. Moreover, loss of program data can not be excluded if the application is terminated improperly.

# Connecting External Accessories

The equivalent USB ports on the front and rear panel of the analyzer can be used to connect a variety of accessories:

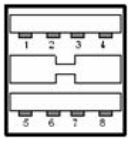
- A mouse simplifies operation of the instrument using the controls and dialogs of the Graphical User Interface (GUI); see *external monitor* below.
- A keyboard simplifies the entry of data.
- A printer generates hard copies of the Graphical User Interface (GUI, see *external monitor* below) and the measurement results.

In addition the analyzer provides interfaces for monitor connection and network integration:

- An external monitor shows the magnified Graphical User Interface (GUI) with all diagram areas and controls. With an additional mouse connected to the analyzer, it is possible to control the analyzer from the the GUI.
- A LAN connection can be established in order to access the hard disk or control the analyzer from an external PC.

## Connecting a Mouse

A USB mouse can be connected to one of the Universal Serial Bus connectors on the front panel or on the rear panel (double USB connectors). It is recommended to use the mouse delivered with the instrument.

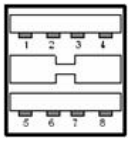


The mouse is detected automatically when it is connected. It is safe to connect or disconnect the mouse during the measurement.

*Use the Start – Control Panel – Mouse menu of Windows XP to configure the mouse properties. To access Windows XP, press the Windows key on the external keyboard or on the hardkey bar of the GUI; see section Connecting a Monitor on p. 1.12.*

## Connecting a Keyboard

A keyboard can be connected to one of the Universal Serial Bus connectors on the front panel or on the rear panel (double USB connectors).

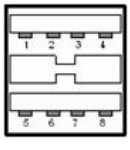


The keyboard is detected automatically when it is connected. The default input language is English – US. It is safe to connect or disconnect the external keyboard during the measurement.

*Use the Start – Control Panel – Keyboard or Regional and Language Options menu of Windows XP to configure the keyboard properties. To access Windows XP, press the Windows key on the external keyboard or on the hardkey bar of the GUI; see section Connecting a Monitor on p. 1.12.*

## Connecting a Printer

A printer can be connected to one of the Universal Serial Bus connectors on the front panel or on the rear panel (double USB connectors).



It is safe to connect or disconnect the printer during the measurement. When printing a copy *(File – Print),* the analyzer checks whether a printer is connected and turned on and whether the appropriate

printer driver is installed. If required, printer driver installation is initiated using Windows XP's *Add Printer Wizard*. The wizard is self-explanatory. A printer driver needs to be installed only once, even though the printer may be temporarily removed from the analyzer.

**Printer driver installation**

A great variety of printer drivers is available on the analyzer. To obtain the complete list, access Windows XP (press the *Windows* key) and open the Add Printer Wizard in the *Start – Control Panel – Printer and Faxes* menu.

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 Printer Wizard* to complete the installation.

Use the *Page Setup* dialog or the *Start – Control Panel – Printers and Faxes* menu of Windows XP to configure the printer properties. To access Windows XP, press the Windows key on the external keyboard or on the hardkey bar of the GUI; see section Connecting a Monitor on p. *1.12*.

## Connecting a Monitor

A standard VGA monitor can be connected to the 15-pole Sub-Min-D MONITOR connector on the rear panel of the analyzer.

**Attention!**

The monitor must be connected while the instrument is switched off (in standby mode). Otherwise correct operation can not be guaranteed.

The monitor displays the magnified Graphical User Interface (GUI) with all diagram areas, measurement results and control elements. No extra configuration is required.

With an additional mouse or keyboard connected to the analyzer, you can control the measurement from the external monitor. The GUI contains the *Hardkey Bar* with key shortcuts for the most important menus and submenus.

## Connecting a LAN Cable

A LAN cable can be connected to one of the LAN connectors on the rear panel of the analyzer. To establish a LAN connection proceed as follows:

1. Refer to section *Assigning an IP Address* and learn how to avoid connection errors.
2. Connect an appropriate LAN cable to one of the LAN ports. Use a commercial RJ-45 cable to establish a non-dedicated network connection, or a cross-over RJ-45 cable to establish a dedicated connection between the analyzer and a single PC.

**Dedicated vs. non-dedicated network connections**

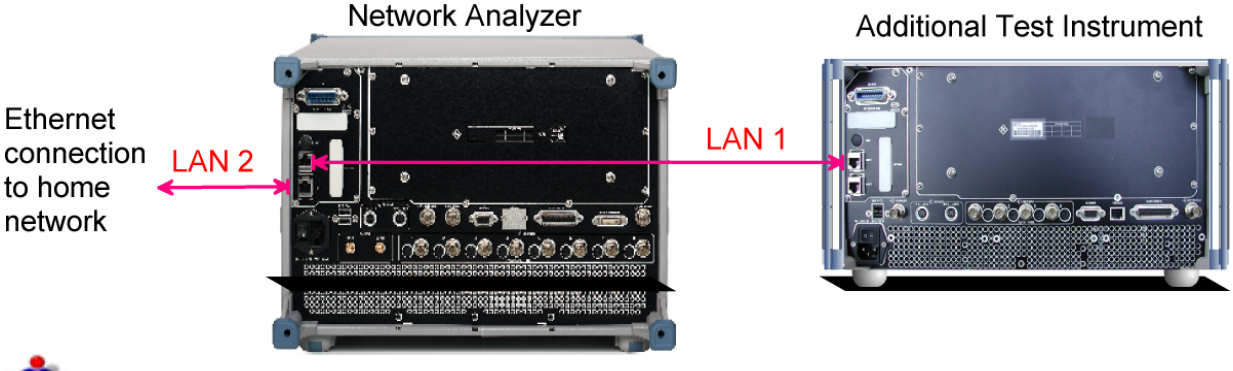
There are two methods to establish a LAN connection of the analyzer:

- A non-dedicated network (Ethernet) connection from the analyzer to an existing network made with an ordinary RJ-45 network cable. The analyzer is assigned an IP address and can coexist with a computer and with other hosts on the same network.
- A dedicated network connection between the analyzer and a single computer made with a cross-over RJ-45 network cable. The computer must be equipped with a network adapter and is directly connected to the analyzer. The use of hubs, switches, or gateways is not needed, however, data transfer is still made using the TCP/IP protocol.

The IP address information is displayed in the *Info – Setup Info* dialog.

## Test Setups with Two LAN Connections

The two LAN connectors on the rear panel of the analyzer are equivalent. With one LAN connector used to establish a connection to a home/company network, the other one can be used to connect an additional instrument, e.g. an additional analyzer or signal generator:




**Defining the network topology: Router vs. network client**

With two LAN connections, it is possible to use the analyzer in two alternative ways:

- As a client participating in two independent networks, one comprising the home network including the analyzer, the second consisting of the additional test instrument plus the analyzer.

- As a data router between the additional test instrument and the home network. This configuration means that the analyzer and the additional test instrument are integrated into a single network.

The network topology is defined in Windows XP's *Control Panel – Network Connections – Local Area Connection Status – Local Area Connection Properties – Internet Protocol (TCP/IP) Properties – Advanced TCP/IP Settings* dialog. Both instruments must have independent IP addresses; see section *Assigning an IP Address* below. Contact your LAN administrator for details.

**Attention!**

Never use both LAN connectors to connect the analyzer in parallel to the same network as this will result in connection errors.

# Remote Control in a LAN

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

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

**Attention!**

An efficient virus protection is a prerequisite for secure operation in the network. Never connect your analyzer to an unprotected network because this may cause damage to the instrument software.

To establish the connection proceed as follows:

1. Press the Windows button and open Windows XP's control panel.
2. Select System and open the Remote tab in the System Properties dialog. Enable Allow users to connect remotely to this computer.
3. Assign an IP address to the analyzer following the directions below and connect the analyzer to the network as described in *Connecting a LAN Cable*.
4. Create a *Remote Desktop Connection* using the analyzer's IP address.

*The analyzer uses a user name and password as credentials for remote access. In the factory configuration,* instrument *is preset for both the user name and the password. To protect the analyzer from unauthorized access, it is recommended to change the factory setting.*

# 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 *alternate TCP/IP* configuration, the addresses must be set manually.

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

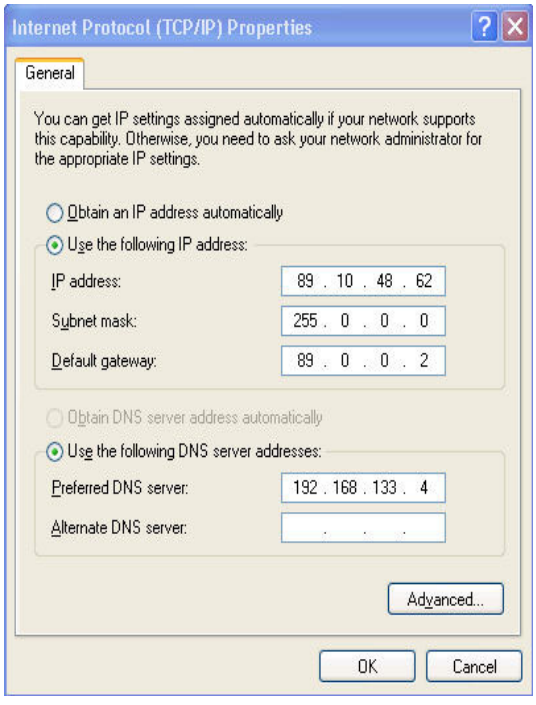
**Attention!**

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

**Manual TCP/IP configuration**

To disable dynamic TCP/IP configuration and enter the TCP/IP address information manually proceed as follows:

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 needed, also obtain the name of your DNS domain and the IP addresses of the DNS and WINS servers on your network. If you use both LAN connectors, you need two different sets of address information.
2. Press the Windows key on the external keyboard or on the hardkey bar of the GUI to access Windows XP (see section *Connecting a Monitor* on p. *1.12)*.
3. Open the *Control Panel – Network Connections – Local Area Connection Status – Local Area Connection Properties – Internet Protocol (TCP/IP) Properties* dialog and enter the address information, e.g.:




For more information refer to the Windows XP Help.

## Remote Desktop Connection

*Remote Desktop* is a Windows application which can be used 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 of the applications, files, and network resources of the analyzer.

To set up a *Remote Desktop* connection

1. Connect the analyzer to a LAN and determine its IP address; see *Remote Control in a LAN.*
2. Set up your remote computer (integrated in the LAN) to use *Remote Desktop* and create the *Remote Desktop* connection to the analyzer.

For detailed information about *Remote Desktop* and the connection refer to the Windows XP Help.

# Windows XP

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

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

**Attention!**

The operating system is adapted to the network analyzer. To avoid impairment of instrument functions, only change the settings described in this manual. Existing software must be modified only with update software released by Rohde & Schwarz. Likewise, only programs authorized by Rohde & Schwarz for use on the instrument must be executed.

The *Start* menu of the operating system is accessed by pressing the Windows key on the external keyboard or on the hardkey bar of the GUI. All necessary settings can be accessed from the *Start* menu, in particular from the *Control Panel.*

# Firmware Update

Upgrade versions of the analyzer firmware are supplied as single setup files *.msi. To perform a firmware update

1. Copy the setup file to any storage medium accessible from the analyzer. This may be the internal hard disk, an external storage medium (USB memory stick, CD-ROM with external drive) or a network connection (LAN, GPIB bus).
2. Start the installation using one of the following methods:
   - If you use an external monitor or PC to display the GUI, access the *Start* menu of the operating system by pressing the Windows key on the external keyboard or on the hardkey bar of the GUI and double-click the setup file.
   - Otherwise, use the remote control command SYSTem:FIRMware:UPDate '<setup_file_name>' to start the installation.
3. Follow the instructions of the setup wizard.

Setup files can be stored and installed again. The default drive name of the internal hard disk is C:. External storage devices are automatically mapped to the next free drive, i.e. D:, E: etc.

**Attention!**

The *Calibration Unit* (accessory R&S ZV-Z51/-Z52 or R&S ZV-Z58) must be disconnected during a firmware update.

# Contents of Chapter 2


**2 Getting Started** ..... 2.1

**Reflection Measurements** ..... 2.2

Instrument Setup for Reflection Measurements ..... 2.2

Parameter and Sweep Range Selection ..... 2.3

Instrument Calibration ..... 2.4

Evaluation of Data ..... 2.6

Saving and Printing Data ..... 2.7

**Basic Tasks** ..... 2.9

Control via Front Panel Keys ..... 2.9

To Access a Particular Menu Command ... ..... 2.9

To Make a Selection in a Dialog... ..... 2.10

Data Entry ..... 2.11

Using Front Panel Keys ..... 2.11

Using the Analyzer's On-Screen Keyboard ..... 2.12

Using Windows XP's On-Screen Keyboard ..... 2.12

Scaling Diagrams ..... 2.13

Setting the Sweep Range ..... 2.13

Reference Value and Position ..... 2.13

Autoscale ..... 2.14

Circular Diagrams ..... 2.14

Using Marker Functions ..... 2.14

Enlarging the Diagram Area ..... 2.15

# 2 Getting Started

The following chapter presents a sample session with a R&S ZVA network analyzer and explains how to solve basic tasks that you will frequently encounter when working with the instrument.

Before starting any measurement on your network analyzer, please note the instructions given in Chapter *Preparing for Use*.

In the *System Overview* you will find detailed information on customizing the instrument and the display according to your personal preferences. For a systematic explanation of all menus, functions and parameters and background information refer to the reference chapters in the online help system.

Use the *Measurement Wizard* in the *System* menu to perform a standard S-parameter measurement 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 different dialogs of the measurement wizard correspond to the typical stages of any measurement:

1. Select the test setup
2. Select the measurement parameters and the diagram areas
3. Define the sweep range
4. Adjust the receiver and source settings (measurement bandwidth, source power)
5. Perform a calibration

*In the following we assume that you are familiar with standard Windows dialogs and mouse operation. Refer to Using Front Panel Keys and Data Entry to learn how to access instrument functions and control dialogs without a mouse and keyboard.*

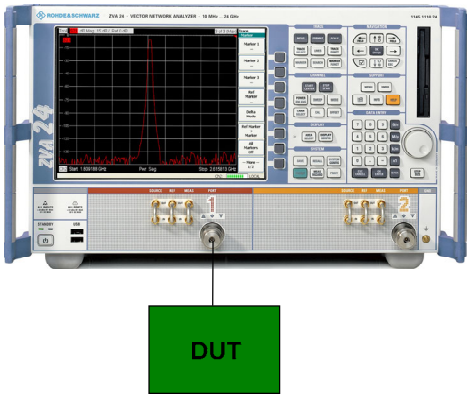
# Reflection Measurements

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. A number of 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 the following example, the analyzer is set up for a reflection measurement, a frequency sweep range and measurement parameter is selected, the instrument is calibrated and the result is evaluated using various formats.

## Instrument Setup for Reflection Measurements

In order to prepare a reflection measurement, you have to connect your DUT (which is assumed to have a male N 50 Ω connector) to one of the (equivalent) analyzer test ports. Besides, it is recommended to preset the instrument in order to set it to a definite, known state.




1. Proceed as described in section *Starting the Analyzer and Shutting Down* to switch on the instrument and start the NWA application.
2. Connect the input port of your DUT to test port 1 of the network analyzer.
3. Press the preset key in the SYSTEM keypad to perform a factory preset of the analyzer.

The analyzer is now set to its default state. The default measured quantity is the transmission S-parameter $S_{21}$. This quantity is zero in the current test setup, so the trace shows the noise level.

Press the TRACE SELECT key in the TRACE keypad and use the softkeys in the *Trace Select* menu if you wish to create a new trace or a new diagram area.

# Parameter and Sweep Range Selection

After preset the display shows a diagram with a dB Mag scale. The sweep range (scale of the horizontal axis) is equal to the maximum frequency range of the analyzer, and the S-parameter $S_{21}$ is selected as a measurement parameter.

To obtain information about the reflection characteristics of your DUT you have to select an appropriate measurement parameter and specify the sweep range.

1. In the CHANNEL keypad, press START CENTER and enter the lowest frequency you want to measure in the *Start Frequency* numeric entry bar (e.g. 5 GHz).

If you use the DATA ENTRY keys at the front panel for data entry, simply type 5 and terminate the entry with the *G/n* key. Refer to section *Data Entry* to learn more about entering numeric values and characters.

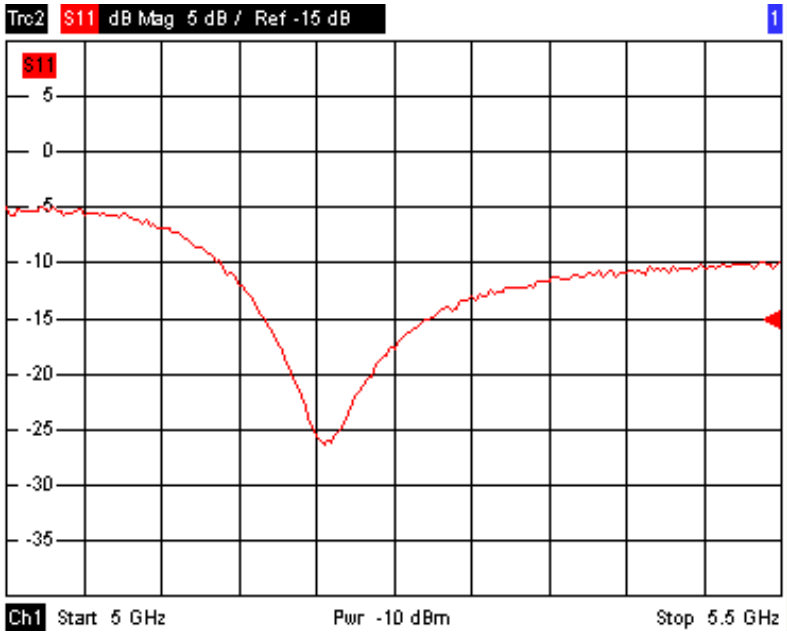
2. Press STOP SPAN and enter the highest frequency you want to measure in the *Stop Frequency* numeric entry bar (e.g. 5.5 GHz).

3. In the TRACE keypad, press MEAS and select the forward reflection coefficient $S_{11}$ as a measurement parameter.

4. In the TRACE keypad, press SCALE and activate the *Autoscale* function.

The analyzer adjusts the scale of the diagram to fit in the entire $S_{11}$ trace, leaving an appropriate display margin.

Refer to section *Scaling Diagrams* to learn more about the different methods and tools for diagram scaling.



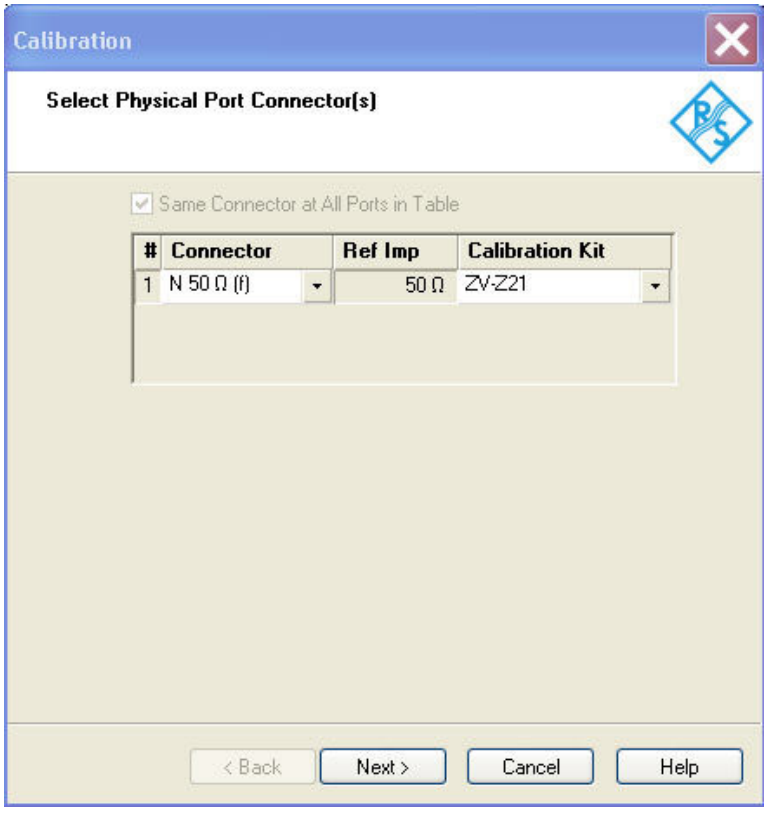
# Instrument Calibration

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.

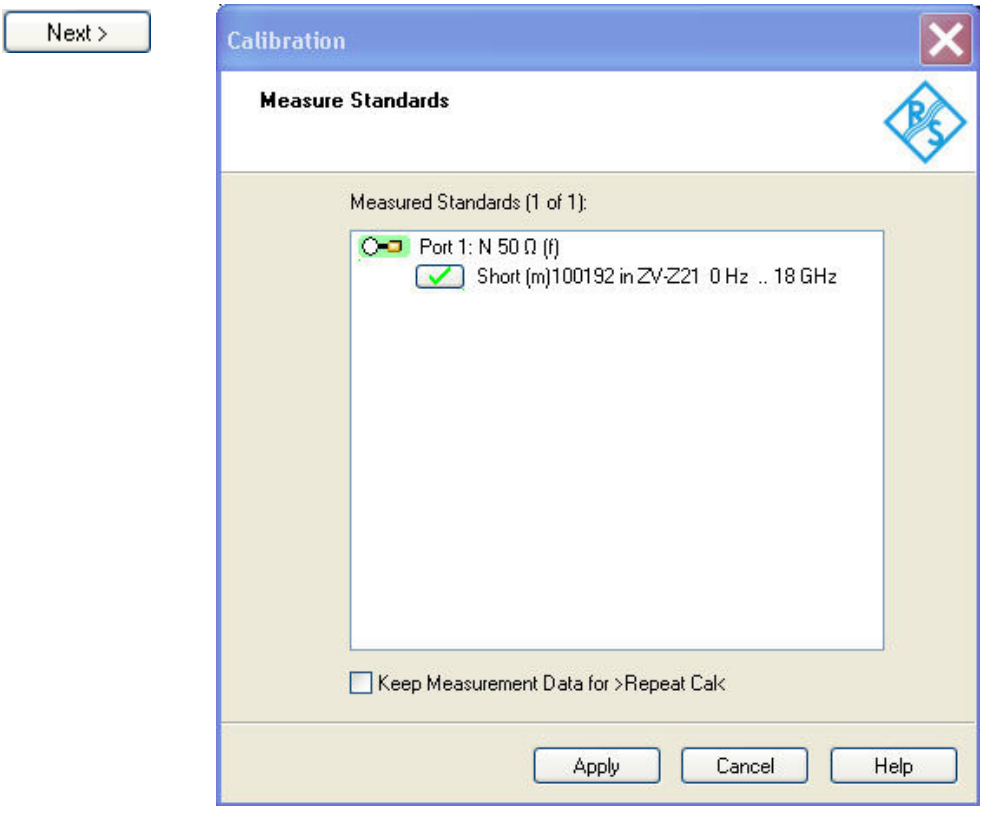
In the following we assume that the calibration kit ZV-Z21 contains an appropriate male short standard with known physical properties. With a single short standard, it is possible to perform a normalization, compensating for a frequency-dependent attenuation and phase shift in the signal path.

Due to the analyzer's calibration wizard, calibration is a straightforward, menu-guided process.

1. Unscrew the DUT and connect the male short standard from calibration kit ZV-Z21.
2. In the CHANNEL keypad, press CAL to open the calibration menu.
3. Activate *Start Cal – One Port P1 – Normalization (Short)* to open the calibration wizard for the selected calibration type.




4. In the first dialog of the wizard, select the calibration kit (here: ZV-Z21) and the test port connector (here: N 50 Ω (f), corresponding to a male calibration standard), and click *Next*.




The next dialog of the calibration wizard shows that only a single calibration standard needs to be measured.

5. Click the box *Short (m)...* to initiate the measurement of the connected short standard.

The analyzer performs a calibration sweep and displays a message box with a progress bar. After completing the sweep the analyzer generates a short sound and a green checkmark appears in the checkbox.

6. Click *Apply* to close the wizard, calculate and store the system error correction data and apply them to the current measurement.
7. Remove the short standard and connect the DUT again.

## Evaluation of Data

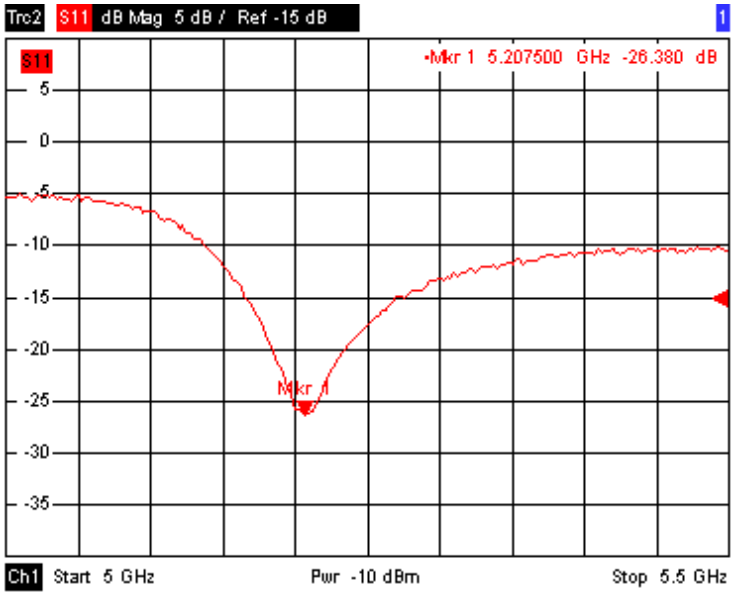
The analyzer provides various tools to optimize the display and analyze the measurement data. For instance, you can use markers determine the maximum of the reflection coefficient, and change the display format to obtain information about the phase shift of the reflected wave and the impedance of your DUT.

1. In the TRACE keypad, press MARKER. This places *Marker 1* to its default position (center of the sweep range).

A marker symbol (triangle) appears on the trace. The stimulus value (frequency) and response value (magnitude of the reflection coefficient converted to a dB value) at the marker position is displayed in the *marker info field* in the upper right corner of the diagram.

2. Press MARKER FUNCT 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.

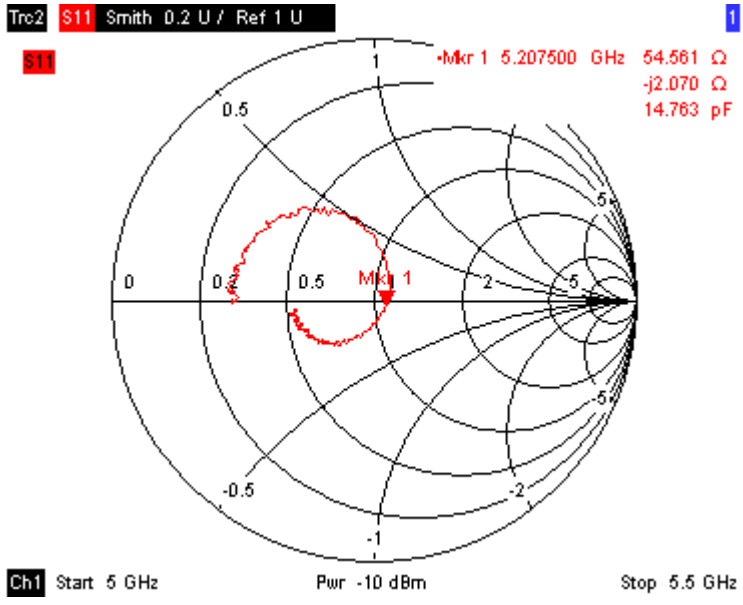


3. In the TRACE keypad, press FORMAT and select the *Phase* of the reflection coefficient to be displayed.

The phase is shown in a Cartesian diagram with a default vertical scale of –225 deg to +225 deg. The *marker info field* shows the frequency and phase at the marker position.

4. Still in the FORMAT menu, select *Smith*.

The Smith chart shows lines of constant real and imaginary part of the impedance in the reflection coefficient plane.




Refer to section *Display Formats and Diagram Types* in Chapter 3 to learn more about the diagram properties.

## Saving and Printing Data

The analyzer provides standard functions for saving measurement settings and for 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 sections *External Accessories* and Remote Control in a LAN to obtain information about the necessary steps.

TRACE FUNCT

1. Press TRACE FUNCT and activate *Import/Export Data – Export.*
2. In the *Export Data* dialog opened, select a file location, format and name and activate *Save.*

The active trace data are written to an ASCII file.

Refer to section *Trace File Formats* to learn more about trace files and their use.

PRINT

3. Press PRINT in the SYSTEM keypad and select *Print Now* to create a hardcopy of your diagram.
4. Select *Print to File...* or *Print to Clipboard* to copy the diagram to a file or an external application.

5. Press SAVE in the SYSTEM keypad.
6. In the *Save As* dialog opened, select a file location, format and name and activate *Save.*

The active setup is stored to a file and can be reused in a later session.

Proceed as described in section *Starting the Analyzer, Shutdown* to shut down your analyzer.

# Basic Tasks

The following sections describe how to solve basic tasks that you will frequently encounter when working with the instrument. In particular you can learn how to access instrument functions and control dialogs without a mouse and keyboard.

## Control via Front Panel Keys

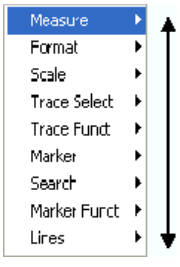
Although a mouse and external keyboard simplify the operation of the instrument, you can access all essential functions using the keys on the front panel. The following examples are intended to make you familiar with front panel key operation.

### To Access a Particular Menu Command ...

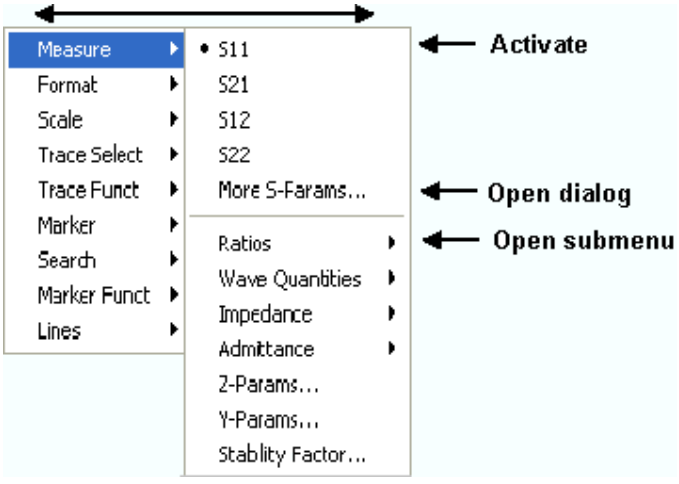
1. Press the MENU key in the SUPPORT keypad to access the menu bar and open the *File* menu.
2. Use the keys in the NAVIGATION keypad or the rotary knob to navigate between and within the menus.

- Use the *Cursor Left* and *Cursor Right* keys to change between the different menus in the menu bar. When the first option in a pull-down menu is a submenu, the submenu will be opened first before proceeding to the next option in the menu bar.

- Use the *Cursor Up and Cursor Down* keys and the rotary knob (if rotated) to scroll up and down in a menu.



- OK ENTER and the rotary knob (if pressed) expand a submenu, open a dialog, or initiate an action, depending on the selected command type.
- CANCEL ESC compresses the current submenu and moves the cursor one menu level up or closes the active dialog, depending on the selected softkey type.



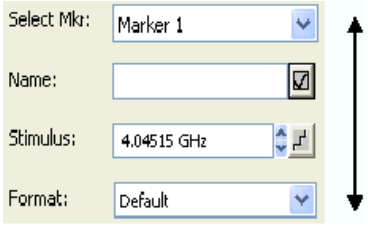

3. As soon as you reach the desired menu command (which must not be one opening a submenu) press OK ENTER or press the rotary knob to initiate an action or open a dialog.

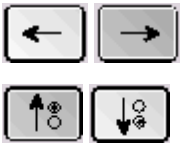
After command execution or after closing the dialog, the menu bar is deactivated and the cursor returns to the diagram/softkey area.

## To Make a Selection in a Dialog...

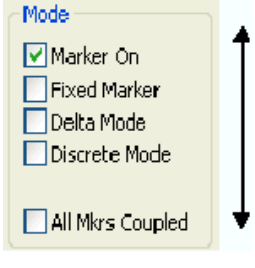
1. Press a softkey with three dots to open a dialog.
2. Use the keys in the NAVIGATION keypad and the rotary knob to access the controls in the dialog.

- Press *Left Field* or *Right Field* to switch between the control elements in a dialog.



- Press the cursor keys or turn the rotary knob to switch between several entries in a list of alternative or independent settings.



3. Use the DATA ENTRY keys or the rotary knob to enter characters and numbers. For more details refer to Data Entry.
4. Press OK ENTER, CANCEL ESC or press the rotary knob to close the active dialog.

## Data Entry

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 alternative ways to enter data.

### Using Front Panel Keys

If no mouse and no external keyboard is connected to the analyzer, you can use the keys in the DATA ENTRY keypad to enter numbers and units and the letters G, M and K.

**To enter a numeric value**

1. Place the cursor into a numeric data input field in a dialog or in the numeric entry bar.
2. Press the DATA ENTRY keys.

- Use *0* to *9* enter the corresponding numbers.

- Use . and - to enter a decimal point or 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 add the appropriate physical unit.

**To enter a character string**

1. Place the cursor into a character data input field in a dialog.
2. Press the DATA ENTRY keys.

- Use *0* to *9* enter the corresponding numbers.

- Use . or - to enter a dot or a hyphen.

- Use *G/n, M/µ,* or *k/m* to enter the letters G, M or K (case-insensitive).

3. To enter letters other than G, M or K, use one of the following methods:

- Turn the rotary knob until the desired letter appears in the character input field.
- If the active input field has a symbol, then use the analyzer's on-screen keyboard.
- Otherwise, use a mouse and Windows XP's on-screen keyboard.

## 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 character input fields which have a symbol.

### Operation with front panel keys

1. Place the cursor into a character data input field in a dialog or in the numeric entry bar.
2. Press OK or the key in the NAVIGATION keypad to open the on-screen keyboard.
3. Use the cursor keys in the NAVIGATION keypad or turn the rotary knob to move the cursor to a character.
4. Press OK ENTER or the rotary knob to select the character for the input string.
5. After completing the input string use the key to move to the *OK* button.
6. Press OK ENTER or the rotary knob to apply your selection and close the keyboard.

### Operation with a mouse

1. Click the symbol to open the on-screen keyboard.
2. Click a sequence of characters and *OK* to apply your selection and close the keyboard.

## Using Windows XP's On-Screen Keyboard

Windows XP's on-screen keyboard allows you to enter characters, in particular letters, even if an input field can not call up the analyzer's own on-screen keyboard. Examples of such fields are the input fields in the *Index* and *Search* tabs of the Help system. A mouse is required for using the Windows XP's on-screen keyboard.

To call up the *On-Screen Keyboard,*

1. Press the Windows key in the SUPPORT keypad to open Windows XP's start menu.
2. Select All Programs – Accessories – Accessibility – On-Screen Keyboard.

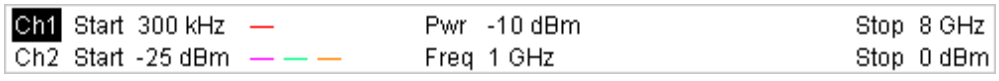
The *System – External Tools* submenu contains a shortcut to Windows XP's on-screen keyboard. Simply click *Mouse Keyboard.lnk* to open the keyboard.

# Scaling Diagrams

The analyzer provides several alternative tools for setting the sweep range and customizing the diagrams. Pick the method that is most convenient for you.

## Setting the Sweep Range

The sweep range for all channels is displayed in the channel list across the bottom of the diagram area:



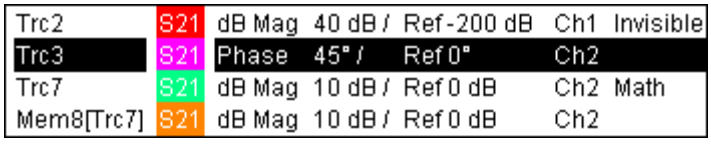

To change the sweep range, use one of the following methods:

- Press the START CENTER or STOP SPAN keys in the CHANNEL keypad.
- Right-click the start or stop value in the channel list and select *Start, Stop, Center, Span* from the context menu.
- Select Start, Stop, Center, Span from the Channel – Stimulus menu.
- Use the marker functions (MARKER FUNCT key).

## Reference Value and Position

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

- Changing the *Reference Value* or *Reference Position* shifts the trace in vertical direction and adjusts the labels of the vertical axis. *Reference 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 *Reference Value* is indicated in the scale section of the trace list.




To change one of the parameters use one of the following methods:

- Press the SCALE key in the TRACE keypad.
- Right-click the scale section in the trace list and select the parameters from the context menu.
- Select the parameters from the *Trace – Scale* menu.
- Use the marker functions (MARKER FUNCT key).

## Autoscale

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

- Press the SCALE key in the TRACE keypad.
- Right-click the scale section in the trace list and select *Autoscale* from the context menu.
- Select *Autoscale* from the *Trace* – *Scale* menu.

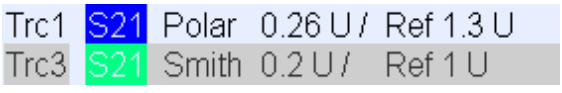
## Circular Diagrams

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

Increasing the *Reference Value* scales down the polar diagram.

Decreasing the *Reference Value* magnifies the polar diagram.

The *Reference Value* is indicated in the scale section of the trace list.



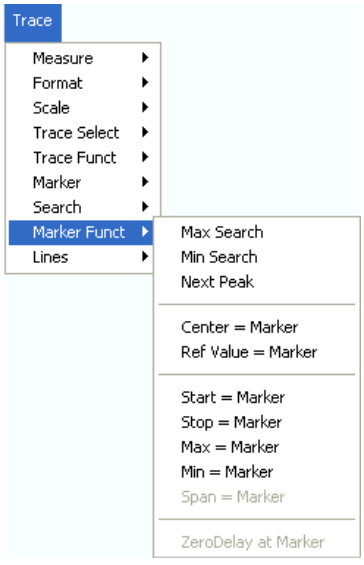

To change the *Reference Value* use one of the following methods:

- Press the SCALE key in the TRACE keypad.
- Right-click the scale section in the trace list and select the parameter from the context menu.
- Select the parameter from the *Trace* – *Scale* menu.
- Use the marker functions.

The *Autoscale* function also works for polar diagrams.

## Using Marker Functions

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. A mouse makes it easier to activate (click) or move (drag and drop) markers.




**To set the sweep range** use one of the following methods:

**Define Start and Stop values**

1. Create two normal markers, e.g. the markers *Mkr 1* and *Mkr 2,* and place them to the desired start and stop values of the sweep range.
2. Activate Mkr 1 and click *Trace – Marker Funct. – Start = Marker*.
3. Activate Mkr 2 and click *Trace – Marker Funct. – Stop = Marker*.

**Use a definite Span**

1. Create a marker and set it to delta mode.

   The analyzer automatically creates a reference marker in addition to the delta marker.
2. Place the reference marker to the desired start value of the sweep range.
3. Set the value of the delta marker equal to the desired (positive or negative) span.
4. Activate the delta marker and click *Trace – Marker Funct. – Span = Marker*.

**To move the trace relative to the vertical axis** proceed as follows:

1. Create a normal marker, e.g. the marker *Mkr 1,* and place it to a particular trace point. E.g. you can use the marker *Search* functions to locate a maximum or minimum on the trace.
2. Click *Trace – Marker Funct. – 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, click *Min = Marker* to move the trace towards the lower diagram edge, or click *Ref Value = Marker* to move the trace towards the Reference Value.

## Enlarging the Diagram Area

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

- *Maximize* allows you to enlarge the active diagram area to occupy the whole window. A double-click on any point in the diagram area is equivalent to the Maximize function.
- The *Title,* the *Softkey Labels,* the *Status Bar* and the *Front Panel Keys* are optional display elements which you can hide in order to gain space for the diagram.

Use the context menu of the diagram area, the keys in the DISPLAY keypad or the *Display* menu to access the scaling functions above.

# Contents of Chapter 3


## 3 System Overview ..... 3.1

**Basic Concepts** ..... **3.1**

Global Resources ..... 3.2

Setups ..... 3.2

Traces, Channels and Diagram Areas ..... 3.2

Trace Settings ..... 3.3

Channel Settings ..... 3.3

**Data Flow** ..... **3.4**

**Screen Elements** ..... **3.6**

Navigation Tools of the Screen ..... 3.6

Menu Bar ..... 3.7

Menu Structure ..... 3.8

Softkey Bar ..... 3.8

Hardkey Bar ..... 3.9

Status Bar ..... 3.9

Display Elements in the Diagram Area ..... 3.10

Title ..... 3.11

Traces ..... 3.11

Markers ..... 3.13

Marker Info Field ..... 3.14

Channel Settings ..... 3.15

Context Menus ..... 3.16

Dialogs ..... 3.16

On-Screen Keyboard ..... 3.17

Step Size ..... 3.18

Numeric Entry Bar ..... 3.18

Display Formats and Diagram Types ..... 3.18

Cartesian Diagrams ..... 3.19

Conversion of Complex into Real Quantities ..... 3.19

Polar Diagrams ..... 3.20

Smith Chart ..... 3.21

Inverted Smith Chart ..... 3.23

Measured Quantities and Display Formats ..... 3.25

**Measured Quantities** ..... **3.26**

S-Parameters ..... 3.26

Multiport Parameters ..... 3.27

Impedance Parameters ..... 3.27

Converted Impedances ..... 3.27

Z-Parameters ..... 3.28

Admittance Parameters ..... 3.29

Converted Admittances ..... 3.29

Y-Parameters ..... 3.30

Wave Quantities and Ratios ..... 3.30

Wave Quantities....3.31
Ratios....3.31
Balance - Unbalance Conversion....3.32
Balanced Port Configurations....3.33
Mixed Mode Parameters....3.34
Mixed Mode Parameters for Different Test Setups....3.34
**Calibration Overview....3.36**
Calibration Standards and Calibration Kits....3.36
Calibration Types....3.37
Normalization....3.38
Full One-Port Calibration....3.38
One-Path Two-Port Calibration....3.38
TOSM Calibration....3.39
TOM Calibration....3.39
TRM Calibration....3.40
TRL Calibration....3.40
TNA Calibration....3.41
Automatic Calibration....3.41
Connecting the Calibration Units to the Network Analyzer....3.42
Performing an Automatic Calibration....3.43
Calibration Procedure....3.44

# 3 System Overview

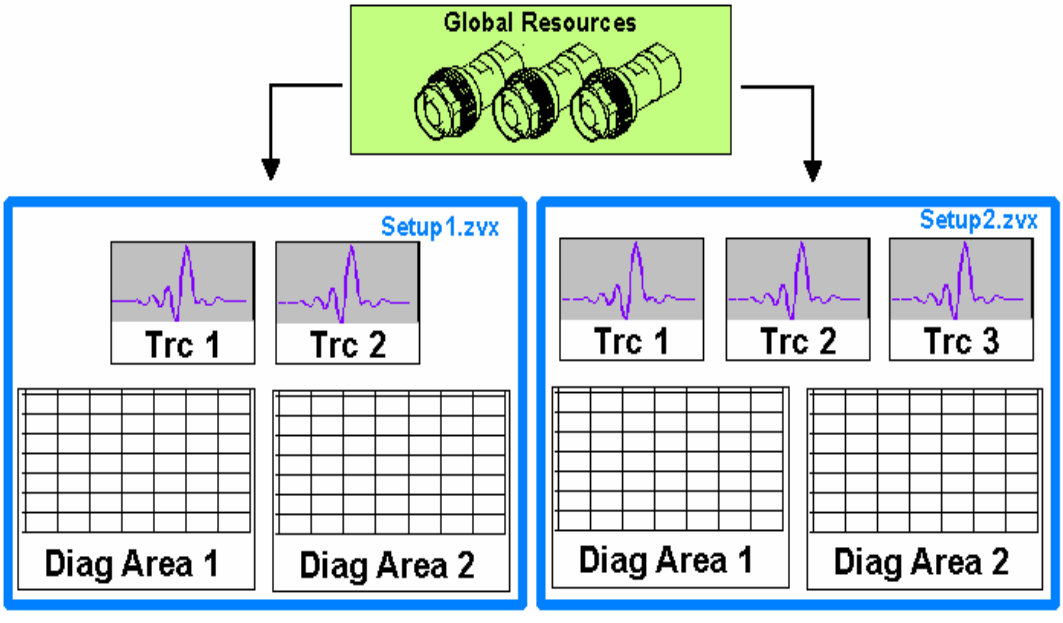
The following chapter provides an overview of the analyzer's capabilities and their use. This includes a description of the basic concepts that the analyzer uses to organize, process and display measurement data, of the screen contents, possible measured quantities, calibration methods and typical test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

For a systematic explanation of all menus, functions and parameters and background information refer to the reference description on the graphical user interface (*GUI Reference)* in the online help system.

## Basic Concepts

The analyzer provides a variety of 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 or setup.
- A *setup* comprises a set of *diagram areas* with all displayed information that can be stored to a setup file.
- The diagram areas show traces which are assigned to channels. See section *Traces, Channels and Diagram Areas*.



# Global Resources

The analyzer provides global settings that are mostly hardware-related and can be used for all measurements, irrespective of the current measurement session or setup. The settings are stored in independent files and do not enter into any of the setup files. The following settings correspond to global resources:

- Calibration kits
- Connector types
- Cal pool
- Color schemes

The data related to global resources are not affected by a *Preset* of the analyzer. However, it is possible to delete or reset global resource data using the *Resets* tab in the *System Config* dialog.

## Setups

A setup comprises a set of *diagram areas* with all displayed information that can be stored to a NWA setup file (*.zvx) and reused. Each setup is displayed in an independent window. The setup file contains the following information:

- General settings related to the setup
- The trace settings for all traces in the diagram areas
- The channel settings for all channels associated to the traces
- The display settings for each diagram area

The *File* menu is used to organize setups.

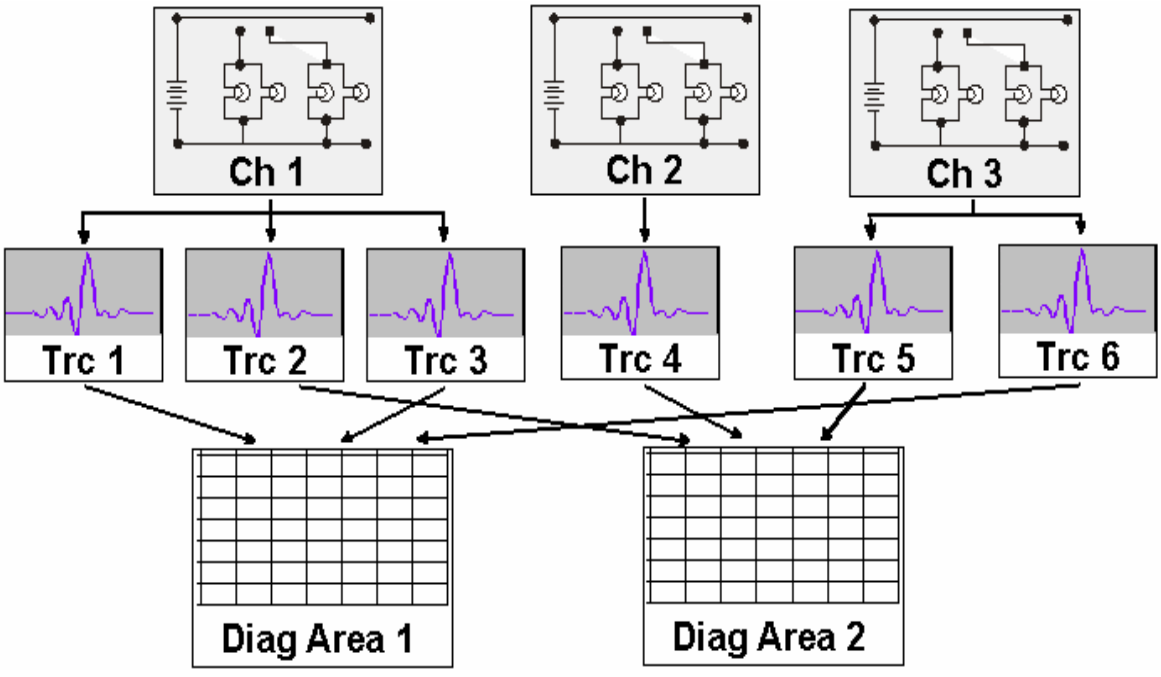
In the *System – External Tools* submenu, you can find demo setups *.vbs for various measurement scenarios. You can modify the demo setups and store them to a *.zvx file for later use. Moreover the *Measurement Wizard* provides predefined, optimized setup s for many measurements.

# Traces, Channels and Diagram Areas

The analyzer arranges, displays or stores the measured data in *traces* which are assigned to *channels* and displayed in *diagram areas*. To understand the menu structure 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 area. The trace settings specify the mathematical operations used in order to obtain traces from the measured or stored data and to display them.
- A channel contains hardware-related settings to specify how the network analyzer collects data.
- A diagram area is a rectangular portion of the screen used to display traces. Diagram areas belonging to the same setup are arranged in a common window. The settings for diagram areas are described in section *Display Elements* in this chapter.

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




## Trace Settings

The trace settings specify the mathematical operations used in order 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* menu provides all trace settings. They complement the definitions of the *Channel* menu. Each trace is assigned to a channel. The channel settings apply to all traces assigned to the channel.

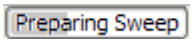
## Channel Settings

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

- Control of the measurement process *(Sweep, Trigger, Average)*
- Description of the test setup *(Power* of the internal source, IF filter *Bandwidth* and *Step Attenuators, Port Configuration)*
- Correction data (Calibration, Offset)

The *Channel* menu provides all channel settings.

*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 visualized by a Preparing Sweep symbol in the status bar:*



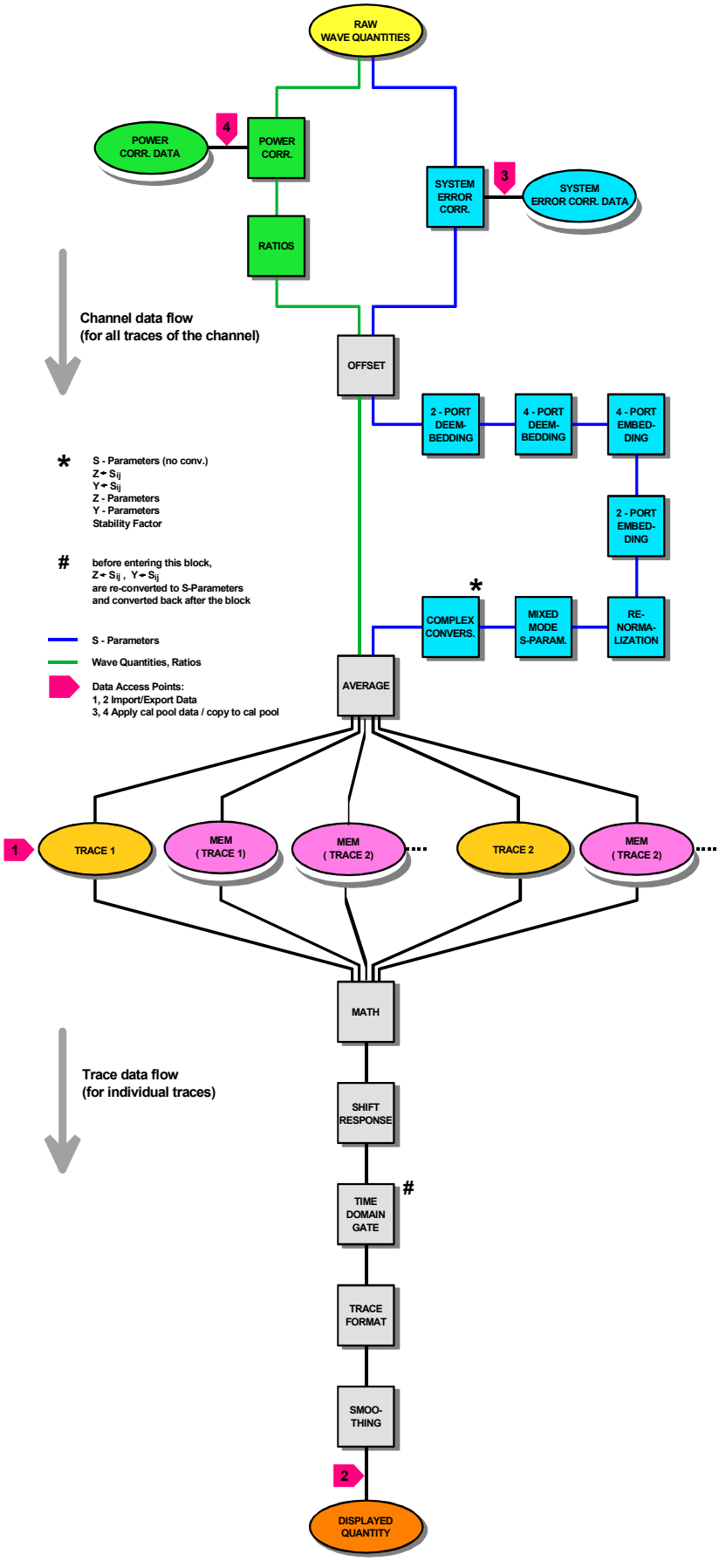

*All analyzer settings can still be changed during sweep initialization. If necessary, the analyzer terminates the current initialization and starts a new preparation period. During the first sweep after a change of the channel settings, an additional red asterisk symbol appears in the status bar:*

# Data Flow

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

The diagram is consists of an upper and a lower part, corresponding to the data processing stages for the entire channel and for the individual traces. All stages in the diagram are configurable. Note that the channel data flow for S-parameters (and quantities derived from S-parameters such as impedances, admittances, stability factors etc.) differs from the channel data flow for wave quantities (and derived quantities such as ratios, PAE etc.).

All stages are described in detail in Chapter 4, *GUI Reference*.


RAW WAVE QUANTITIES
POWER CORR. DATA
4
POWER CORR.
RATIOS
SYSTEM ERROR CORR.
3
SYSTEM ERROR CORR. DATA
Channel data flow (for all traces of the channel)
OFFSET
2 - PORT DEEM-BEDDING
4 - PORT DEEM-BEDDING
4 - PORT EMBED-DING
2 - PORT EMBED-DING
*
S - Parameters (no conv.)
Z ⇽ Sij
Y ⇽ Sij
Z - Parameters
Y - Parameters
Stability Factor
#
before entering this block, Z ⇽ Sij , Y ⇽ Sij are re-converted to S-Parameters and converted back after the block
S - Parameters
Wave Quantities, Ratios
Data Access Points:
1, 2 Import/Export Data
3, 4 Apply cal pool data / copy to cal pool
COMPLEX CONVERS.
MIXED MODE S-PARAM.
RE-NORMA-LIZATION
AVERAGE
1
TRACE 1
MEM ( TRACE 1)
MEM ( TRACE 2)
TRACE 2
MEM ( TRACE 2)
MATH
Trace data flow (for individual traces)
SHIFT RESPONSE
TIME DOMAIN GATE
#
TRACE FORMAT
SMOO-THING
2
DISPLAYED QUANTITY

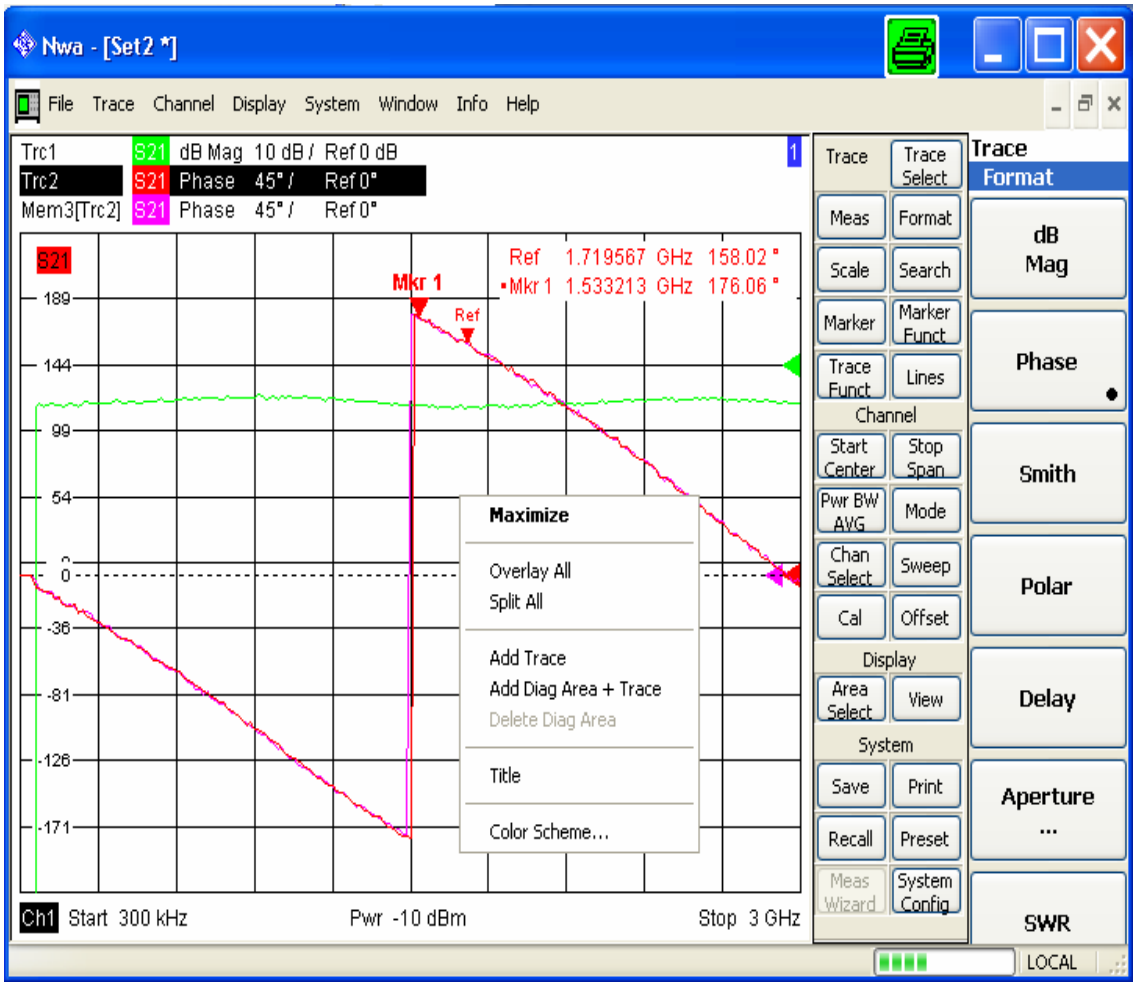
# Screen Elements

This section describes the operating concept of the network analyzer, including the alternative navigation tools for mouse and hardkey operation, the trace settings, markers and diagram areas. For a description of the different quantities measured by the analyzer refer to the *Measured Quantities* section.

## Navigation Tools of the Screen

The main window of the analyzer provides all control elements for the measurements and contains the diagram areas for the results. There are several alternative ways to access an instrument function:

- Using the menus and submenus of the menu bar (provides all settings)
- Using the softkeys of the softkey bar (alternative to the previous method)
- Using the hardkey bar (preselection of the most important menus)




- Refer to section *Display Elements* to obtain information about the results in the diagram area.
- Refer to section *Display Menu* in the reference chapter and learn how to customize the screen.

## Menu Bar

All analyzer functions are arranged in drop-down menus. The menu bar is located across the top of the diagram area:

Menus can be controlled in different ways:

- With a mouse, like the menus in any Windows application. A left mouse click expands a menu or submenu. If a menu command has no submenu assigned, a left mouse click opens a dialog or directly activates the menu command.
- Using the front panel keys.
- With a combination of the previous methods, using the hardkey bar (front panel key bar, activated via *Display – Config./View)*.

The active menu is the menu containing the last executed command. If the softkey bar or hardkey bar is displayed *(Display – Config./View – Softkey Labels* on), then the active menu is indicated on top of the softkey/hardkey bar.

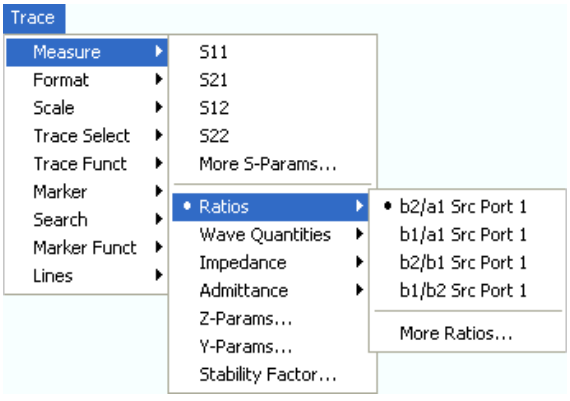
When you select a command in a new menu the softkey bar is updated to reflect the new active menu with all commands. You can continue operation using the softkeys.

**Ove**rview of menu functions

- The *Control* menu provide standard Windows functions to control windows.
- The *File* menu provides standard Windows functions to create, save, recall or print setups, to copy the current screen and to shut down the application.
- The *Trace* menu provides all trace settings and the functions to create, select, modify and store different traces. In addition the menu provides the marker, search and limit check functions.
- The *Channel* menu provides all channel settings and the functions to create, select, modify and store different channels.
- The *Display* menu provides all display settings and the functions to create, select, modify and arrange different diagram areas.
- The *System* menu provides functions to reverse operations, return to a defined instrument state, retrieve information on the instrument and access service functions. Besides, it configures the remote control operation, starts the *Measurement Wizard* and provides print options.
- The *Window* menu provides standard Windows™ functions to arrange different windows on the screen.
- The *Info* menu provides information and selftest options for service purposes and troubleshooting.
- The *Help* menu provides assistance with the network analyzer and its operation.

## Menu Structure

All menus show an analogous structure.

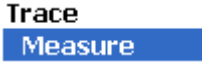



- A menu command with a right arrow ▸ expands a submenu with further related settings.
  **Example:** *Measure* ▸ expands a submenu to select the quantity to be measured and displayed.
- A menu command with three dots appended calls up a dialog providing several related settings.
  **Example:** *More S-Parameters...* opens a dialog to select S-parameters for multiport measurements or mixed mode S-parameters.
- A menu command with no arrow or dots directly initiates an action.
  **Example:** *S21* selects the forward transmission coefficient $S_{21}$ as measured quantity.
- A dot preceding the menu command indicates the current selection in a list of alternative settings.
  **Example:** In the figure above, $S_{21}$ is selected as measured quantity.

## Softkey Bar

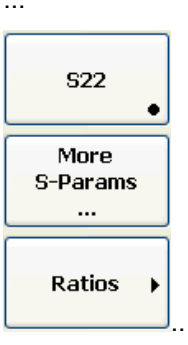
The softkey bar displays the commands of the active menu so that they can be activated by hitting the associated keys on the front panel. It consists of three elements:

Heading




The heading shows the two lowest level menu commands in the current branch of the menu tree. The lowest-level command appears on a shaded background.

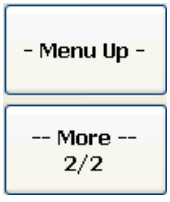
Function softkeys




Up to 8 softkeys, each corresponding to a command of the active menu. The function of the softkeys and their labels are strictly equivalent to the corresponding menu commands.

- A large dot in the lower right corner indicates the current selection in a list of alternative settings.
- Three dots indicate that the softkey calls up a dialog providing several related settings.
- A right arrow ▸ indicates a submenu with further related settings.
- A softkey with no arrow or dots directly initiates an action.

Navigation softkey (optional)




Softkey no. 8 or softkeys no. 7 and no. 8 are reserved for navigation:

- *More 1/2* and *More 2/2* toggle between two groups of softkeys which belong to the same menu. The softkeys are provided whenever the active menu contains more than 7 commands.
- *Menu Up* activates the higher-level menu and is provided in all menus except the top-level one listing the main menus in the menu bar.

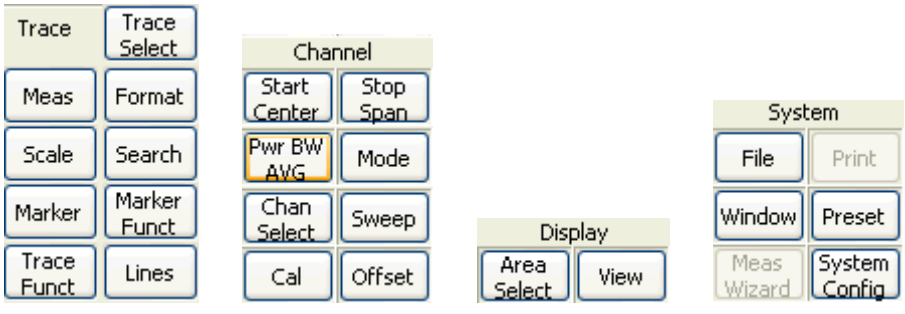
The softkey bar is automatically updated when the active menu is changed.

TIPS You can hide the softkey bar and gain screen space for the diagram areas if you use a mouse to control the analyzer (see *Display – Config./View*). All settings are accessible from the menus listed in the menu bar across the top of the screen. Moreover, you don't have to display the softkey bar permanently in order to make use of its functionality. Hitting any of the keys associated to the softkey bar will make it visible for a period of time sufficient to select the next instrument function.

## Hardkey Bar

The hardkey bar (front panel key bar, *Display – Config./View)* represents the most commonly used front panel keys of the analyzer. Clicking a key symbol executes the action of the corresponding key.

The hardkey bar corresponds to the TRACE, CHANNEL, DISPLAY and SYSTEM keypads:

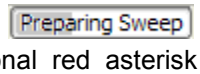



TIPS The hardkey bar provides access to the basic groups of settings with a single mouse click. It is particularly useful if the analyzer is controlled from an external monitor or *Remote Desktop*. Alternatively the settings are accessible from the menus of the menu bar or from the softkey bar.

The hardkey bar is hidden by default to gain screen space for the diagram areas.

## Status Bar

The status bar (*Display – Config./View)* describes the current channel (if the setup contains only one channel), the progress of the sweep and the control mode of the analyzer *(LOCAL* or *REMOTE)*.

During sweep initialization, the progress bar for the sweep is replaced by a 

symbol. During the first sweep after a change of the channel settings, an additional red asterisk symbol appears:

TIP You can hide the status bar and gain screen space for the diagram areas.

# Display Elements in the Diagram Area

The central part of the screen is occupied by one or several diagram areas.

Diagram Areas

A diagram area is a rectangular portion of the screen used to display traces. Diagram areas are arranged in windows; they are independent of trace and channel settings. A diagram area can contain a practically unlimited number of traces, assigned to different channels (overlay mode).

Diagram areas are controlled and configured by means of the functions in the *Display* menu and the following additional settings:

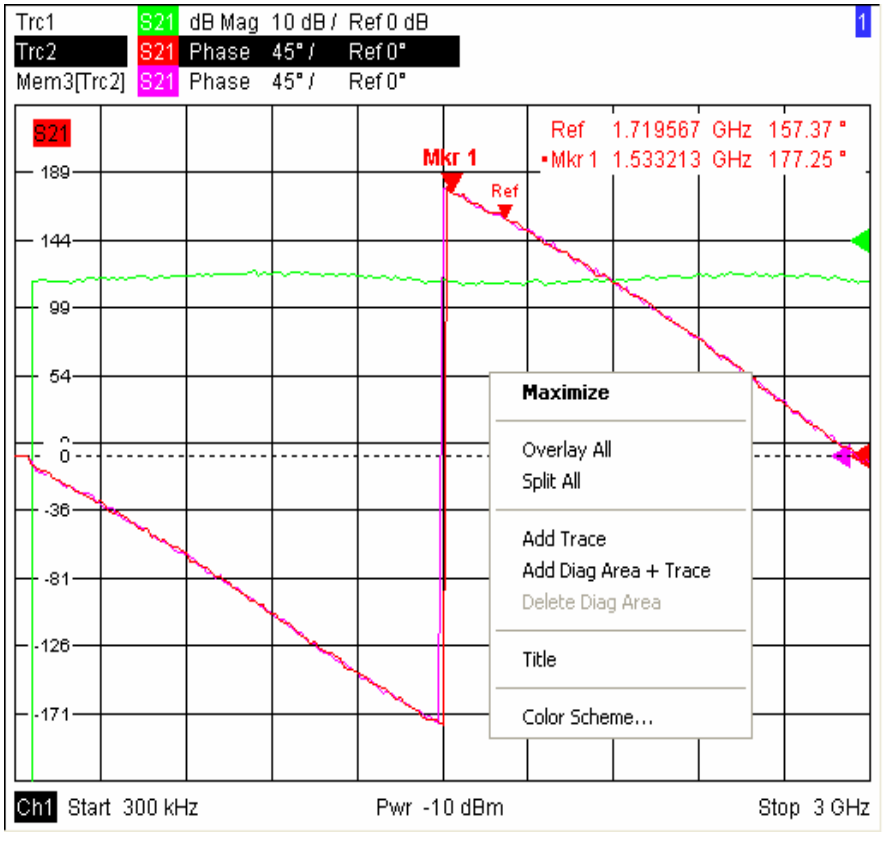
- The settings in the *Window* menu arrange several windows containing one or more diagram areas within the entire screen. Each window corresponds to a setup. Only one setup can be active at a time, and only the traces of the active setup are updated by the current measurements.
- Various settings to assign traces to diagram areas are provided in the *Trace* – *Traces* submenu.

Diagram areas may contain:

- Measurement results, in particular the traces and marker values
- An indication of the basic channel and trace settings
- Context menus providing settings related to the current screen

NOTE

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



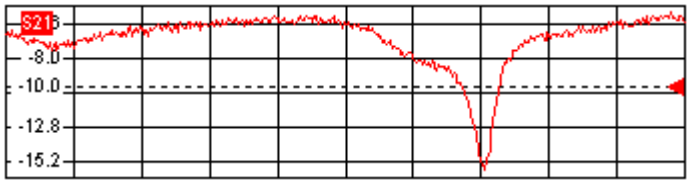
## Title

Across the top of the diagram area, an optional title describes the contents of the area. Different areas within a setup are distinguished by area numbers in the upper right corner.

Use the context menu or the functions in the *Display* menu to display, hide or change the title and to add and customize diagram areas.

## Traces

A trace is a set of data points displayed together in the diagram area. 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 in order to modify the diagram scale and shift the trace vertically.
- ***Measured quantity*** (for the active trace): The measured quantity is indicated in the left upper corner of the diagram.

A trace can be either a data trace, a memory trace, or a mathematical trace; see *Trace Types* below.

Context menu of the diagram area

A right mouse click on any point of the diagram area (except the marker info field and the measured quantity info) opens a context menu:

The settings correspond to the most common commands in the *Display – Area Select* and *Display – Config View* menus.

## Trace Types

The analyzer uses traces to display the current measurement result in a diagram area but 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:

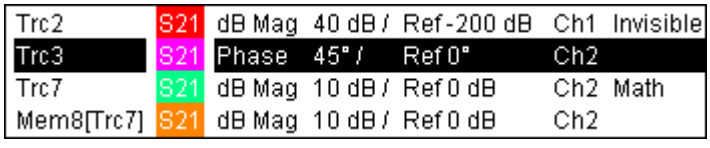
- **Data traces** show the current measurement data and are continuously updated as the measurement goes on. Data traces are dynamic traces.
- **Memory traces** are generated by storing the data trace to the memory. It represents the state of the data trace at the moment when it was stored. Memory traces 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 setup. 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, see *Data -> Mem*. Markers and marker functions are available for all trace types.

The trace type of each trace in a diagram area is indicated in the trace list. You can also make each trace *Invisible* without deleting it.

## Trace List and Trace Settings

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



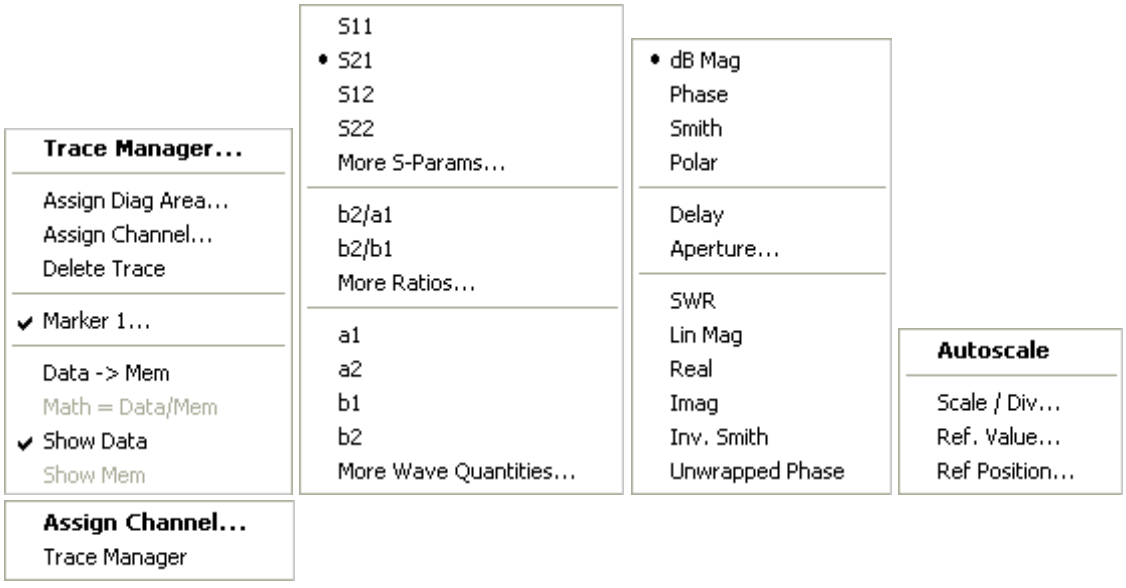

Each line in the trace list describes a single trace. The active trace is highlighted. The lines are divided into several sections with the following contents (from left to right):

- **Trace name** indicates the current trace name. The default names for new traces are Trc<n> where <n> is a current number. A *Mem...* preceding the trace name indicates a memory trace. Right-click the section and call the *Trace Manager* from the context menu to change the trace name.
- **Measured quantity** indicates the measured quantity, e.g. an S-parameter or a ratio. The measured quantity of the active trace is also displayed in the diagram area below the trace list.
- **Format** shows how the measured data is presented in the graphical display (trace format).
- **Scale** shows the value of the vertical or radial diagram divisions (Scale Div.) and the Reference Value.
- **Channel** shows the channel that each trace is assigned to. The channel section is omitted if the all traces in the diagram area are assigned to the same channel.
- **Type** shows *Invisible* if a trace is hidden and *Math* if the trace is a mathematical trace. Right-click the trace name and click *Show Data* or *Show Mem* from the context menu to display and hide data and memory traces. Use the *Trace Funct(ions)* to define mathematical traces.

Right-click any of the sections in the trace list (except *Type)* to open a context menu and access the most common tasks related to the section.

Context menus of the trace list

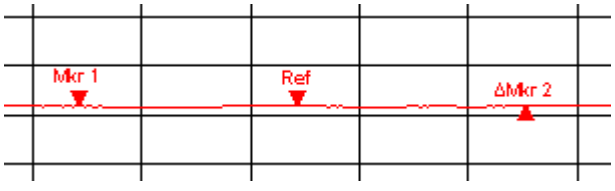
A right mouse click on the trace name, the measured quantity, and the format and scale section of the trace list opens the following context menus, respectively:




The settings correspond to the most common commands in the *Trace – Trace Select, Trace – Trace Funct, Trace – Meas, Trace – Format and Trace – Scale* menus.

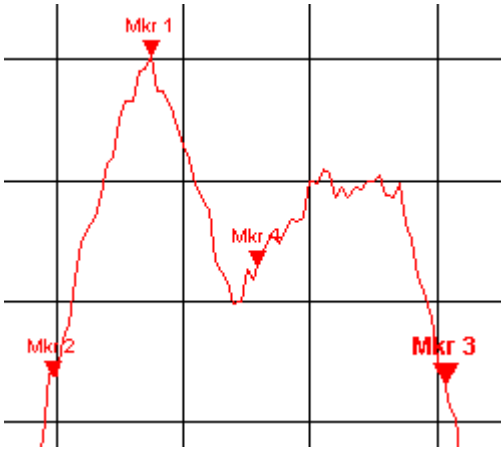
## Markers

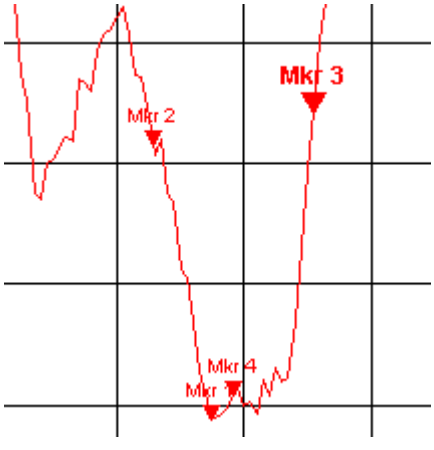
Markers are tools for selecting points on the trace and for numerical readout of measured data. The analyzer provides three different basic marker types.




- A (normal) marker *(Mkr 1, Mkr 2, ...)* determines the coordinates of a measurement point on the trace. Up to 10 different normal markers can be assigned to a trace.
- A reference marker *(Ref)* defines the reference value for all delta markers.
- A delta marker *(Δ)* indicates the coordinates relative to the reference marker.
- The stimulus value of a discrete marker always coincides with a sweep point so that the marker does not show interpolated measurement values.

The markers 1 to 4 are also used for bandfilter search mode. The examples below show a bandpass search and a bandstop search, respectively.

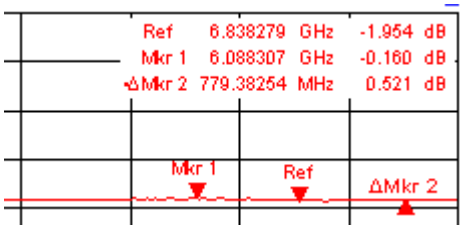


- *Mkr 1* indicates the maximum (minimum) of the peak.
- *Mkr 2* and *Mkr 3* indicate the lower and upper band edge where the trace value has decreased (increased) by a definite *Level* value.
- *Mkr 4* indicates the center of the peak, calculated as the arithmetic mean value of the *LBE* and *UBE* positions.

## Marker Info Field

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




The list contains the following information:

- *Mkr 1, Mkr2,* ... 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 *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.
- A Δ sign placed in front of the marker line indicates that the marker is in *Delta Mode.*

**Customizing the marker info field**

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

- Double-click the info field to open the *Mkr Properties* dialog with extended settings for all markers of the active trace.
- Right-click the info field to open a context menu providing frequently used marker settings.
- To change the position of the marker info field, select *Movable Marker Info* from the context menu. Drag-and-drop the info field to any position in the active diagram area.
- To change the format of the active marker, select *Mkr Format.*
- To express the coordinates of the active marker relative to the reference marker, activate the *Delta Mode.*

For more information: Show Info Table

In addition to the marker info field, the analyzer provides an info table with extended marker information.

| Marker | Trace | Stimulus | Response | Delta | Discr | Fixed | Tracking | Search Range |
|---|---|---|---|---|---|---|---|---|
| Ref | Trc1 | 3.440171000 GHz | -6.426 dB | ☐ | ☑ | ☑ | Off | Full Range |
| Mkr 1 | Trc1 | 4.000150000 GHz | -5.364 dB | ☐ | ☐ | ☐ | Off | Full Range |

The table is hidden by default. To display the table double-click the marker info field to open the *Marker Properties* dialog.

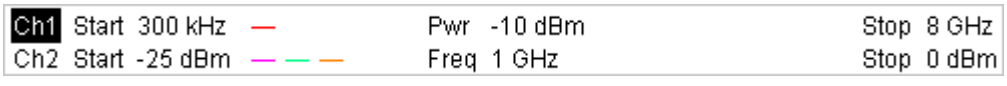
Context menu of the marker info field

A right mouse click on the marker info field opens a context menu:

*Movable Marker Info* allows the marker info field to be placed to any position in the diagram area. The remaining settings correspond to the most common commands in the *Trace – Marker* and *Trace – Search* menus.

## Channel Settings

The main properties of all channels assigned to the traces in the diagram area 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):

- **Channel name** indicates the current channel name. The default names for new channels are Ch<n> where <n> is a current number. Right-click the section and call the *Channel Manager* from the context menu to change the channel name.
- **Start value of the sweep** indicates the lowest value of the sweep variable (e.g. the lowest frequency measured), corresponding to the left edge of the Cartesian diagram.
- **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.
- **Additional stimulus parameter** shows either the power of the internal signal source (for frequency sweeps and time sweeps) or the CW frequency (for power sweeps).
- **Stop value of the sweep** indicates the highest value of the sweep variable (e.g. the highest frequency measured), corresponding to the right edge of the Cartesian diagram.

Right-click any of the sections in the trace list (except *Color legend)* to open a context menu and access the most common tasks related to the section.

Context menus of the channel list

A right mouse click on the channel name, the sweep range, and the additional parameter section of the channel list opens the following context menus, respectively:

The settings correspond to the most common commands in the *Channel – Channel Select, Channel – Stimulus* and *Channel – Power Bandwidth Average* menus.

### 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 area
- 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)

Working with context menus requires a mouse. Click inside the display element that you want to work with using the **right** mouse button.

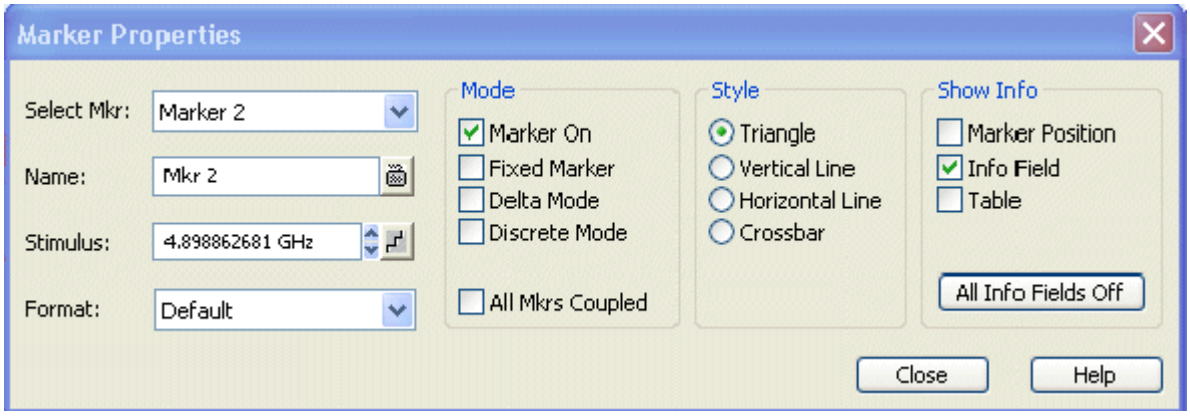
Except from some particular screen configurations, anything you can do from a context menu you can also do from the menu bar or using front panel keys and softkeys. Use whatever method is most convenient for you.

## 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 *Marker Properties...*) call up a dialog. The dialogs of the analyzer have an analogous structure and a number of common control elements.

The *Dialog Transparency* function in the *System Config* menu varies the transparency of all dialogs. With an appropriate setting, you can control the dialogs and at the same time view the underlying traces and display elements.

*We assume that you are familiar with standard Windows dialogs and mouse operation. Refer to Using Front Panel Keys to learn how to control dialogs without a mouse and keyboard.*




## Immediate vs. Confirmed Settings

In some dialogs, the settings take effect immediately so that the effect on the measurement is observable while the dialog is still open. This is especially convenient when a numeric value is incremented or decremented, e.g. via the rotary knob.

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: *Step Size* dialog.
- Dialogs with confirmed settings provide both an OK button and a *Cancel* button. Example: On-screen keyboard.

You can also cancel an immediate setting using *System – Undo!*.

## On-Screen Keyboard

A keyboard symbol next to a character input field opens the analyzer's on-screen keyboard.

The on-screen keyboard contains two sets of characters plus the following additional controls:

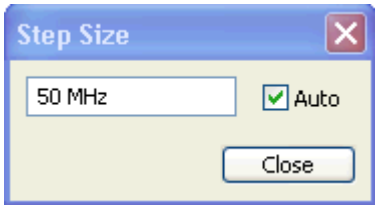
- *Shift* changes between the two character sets containing lower case letters/numbers and upper case letters/special characters, respectively.
- *<= BS* deletes the current string in the alphanumeric input field.
- *OK* applies the current selection and closes the keyboard. The current string is written into the input field of the calling dialog. See also *Immediate vs. Confirmed Settings*.

- *Cancel* discards the current selection and closes the keyboard. The input field of the calling dialog is left unchanged.

The on-screen keyboard allows you to enter characters, in particular letters, without an external keyboard; see *Data Entry.* To enter numbers and units, you can also use the DATA ENTRY keys on the front panel of the instrument.

### Step Size

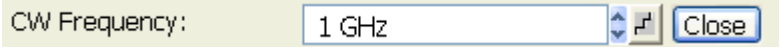
A step symbol next to a numeric input field opens the *Step Size* dialog to define an increment for data variation using the *Cursor Up/Down* buttons in the dialogs or the rotary knob.




- The input value for the step size takes effect immediately; see *Immediate vs. Confirmed Settings.*
- *Auto* activates the default step size for the current input parameter.
- *Close* closes the *Step Size* dialog.

### Numeric Entry Bar

Single numeric values can be entered using the input field of the numeric entry bar. The numeric entry bar appears just below the menu bar as soon as a function implying a single numeric entry is activated. In contrast to dialogs, it does not hide any of the display elements in the diagram area.




The numeric entry bar contains the name of the calling function, a numeric input field including the Cursor Up/Down buttons for data variation and a step symbol, and a *Close* button. Besides it is closed automatically as soon as an active display element in the diagram area is clicked or a new menu command is activated.

## Display Formats and Diagram Types

A display format defines how the set of (complex) measurement points is converted and displayed in a diagram. The display formats in the *Trace – Format* menu use the following basic diagram types:

- Cartesian (rectangular) diagrams are used for all display formats involving a conversion of the measurement data into a real (scalar) quantity, i.e. for *dB Mag, Phase, Delay, SWR, Lin Mag, Real, Imag* and *Unwrapped Phase.*
- Polar diagrams are used for the display format *Polar* and show a complex quantity as a vector in a single trace.
- Smith charts are used for the display format *Smith*. They show a complex quantity like polar diagrams but with grid lines of constant real and imaginary part of the impedance.

- Inverted Smith charts are used for the display format *Inverted Smith*. They show a complex quantity like polar diagrams but with grid lines of constant real and imaginary part of the admittance.

NOTE

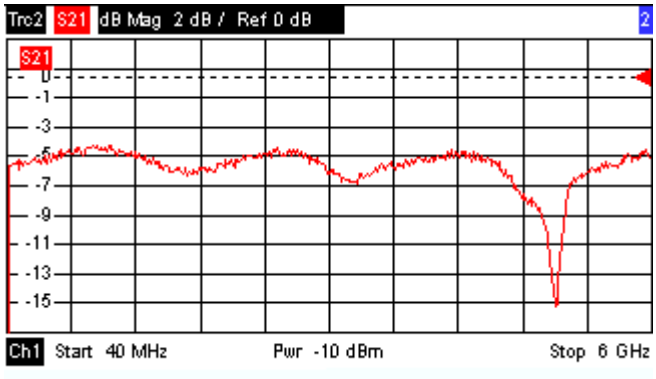
*The analyzer allows arbitrary combinations of display formats and measured quantities (Trace – Measure). Nevertheless, in order to extract useful information from the data, it is important to select a display format which is appropriate to the analysis of a particular measured quantity; see Measured Quantities and Display Formats.*

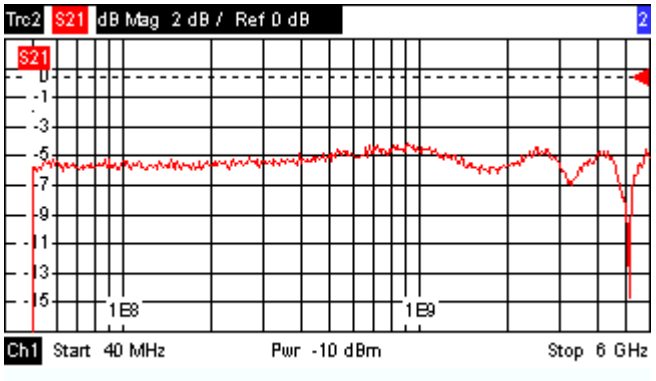
## Cartesian Diagrams

Cartesian diagrams are rectangular diagrams used to display a scalar quantity as a function of the stimulus variable (frequency / power / time).

- The stimulus variable appears on the horizontal axis (x-axis), scaled linearly (sweep types *Lin Frequency, Power, Time, CW Mode)* or logarithmically (sweep type *Log Frequency)*.
- The measured data (response values) appears on the vertical axis (y-axis). The scale of the y-axis is linear with equidistant grid lines although the y-axis values may be obtained from the measured data by non-linear conversions.

The following examples show the same trace in Cartesian diagrams with linear and logarithmic x-axis scaling.




## Conversion of Complex into Real Quantities

The results to be selected in the *Trace – Measure* menu can be divided into two groups:

- S-Parameters, Ratios, Wave Quantities, Impedances, Admittances, Z-Parameters and Y-Parameters are complex.
- *Stability Factors* and *DC Input* values (voltages, PAE) are real.

The following table shows how the response values in the different Cartesian diagrams are calculated

from the complex measurement values z = x + jy (where x, y, z are functions of the sweep variable). The formulas also hold for real results, which are treated as complex values with zero imaginary part (y = 0).

| Trace Format | Description | Formula |
|---|---|---|
| dB Mag | Magnitude of z in dB | $\|z\| = \sqrt{x^2 + y^2}$<br>dB Mag(z) = 20 * log\|z\| dB |
| Lin Mag | Magnitude of z, unconverted | $\|z\| = \sqrt{x^2 + y^2}$ |
| Phase | Phase of z | $\phi(z) = \arctan(y/x)$ |
| 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\|) |
| Delay | Group delay, neg. derivative of the phase response | $-d\phi(z) / d\omega$ ($\omega = 2\pi * f$) |

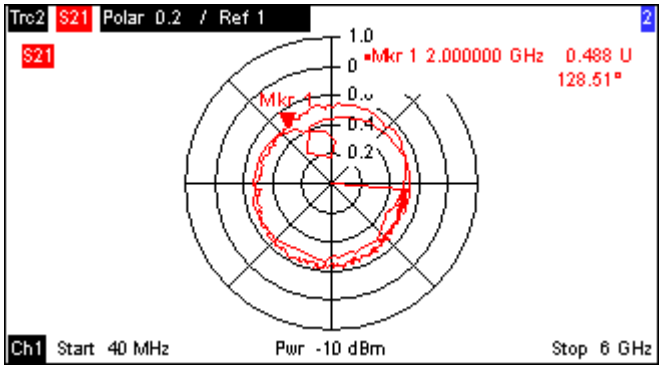
An extended range of formats and conversion formulas is available for markers. To convert any point on a trace, create a marker and select the appropriate marker format. Marker and trace formats can be selected independently.

## Polar Diagrams

Polar diagrams show the measured data (response values) in the complex plane with a horizontal real axis and a vertical imaginary axis. The grid lines correspond to points of equal magnitude and phase.

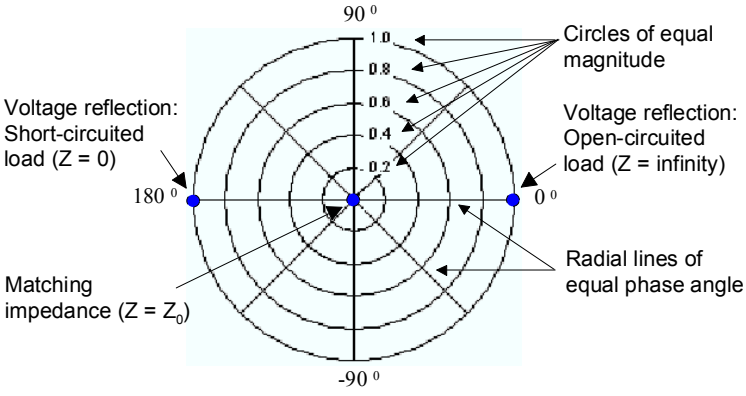
- The magnitude of the response values corresponds to their distance from the center. Values with the same magnitude are located on circles.
- The phase of the response values is given by the angle from the positive horizontal axis. Values with the same phase on straight lines originating at the center.

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), whereas the outer circumference ($|S_{ii}| = 1$) represents a totally reflected signal.




Examples for definite magnitudes and phase angles:

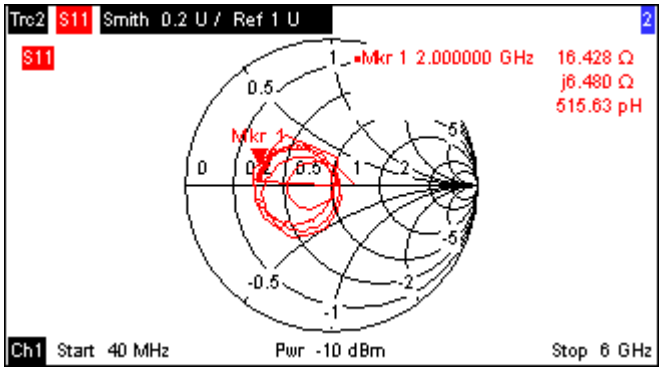
- The magnitude of the reflection coefficient of an open circuit (Z = infinity, I = 0) is one, its phase is zero.
- The magnitude of the reflection coefficient of a short circuit (Z = 0, U = 0) is one, its phase is $-180^0$.

## Smith Chart

The Smith chart is a circular diagram that maps the complex reflection coefficients $S_{ii}$ to normalized impedance values. In contrast to the polar diagram, the scaling of the diagram is not linear. The grid lines correspond to points of constant resistance and reactance.

- Points with the same resistance are located on circles.
- Points with the same reactance produce arcs.

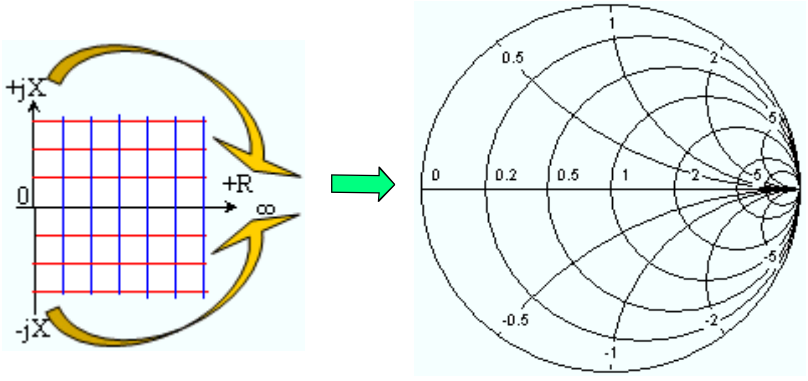
The following example shows a Smith chart with a marker used to display the stimulus value, the complex impedance Z = R + j X and the equivalent inductance L (see marker format description in the help system).




TIP A comparison of the Smith chart, the inverted Smith chart and the polar diagram reveals many similarities between the two representations. In fact the shape of a trace does not change at all if the display format is switched from *Polar* to *Smith* or *Inverted Smith* – the analyzer simply replaces the underlying grid and the default marker format.

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 $\Gamma$ (e.g. $S_{11}$, $S_{22}$), then the unit Smith chart can be used to read the normalized impedance of the DUT. The coordinates in the normalized impedance plane and in the reflection coefficient plane are related as follows (see also: definition of matched-circuit (converted) impedances):

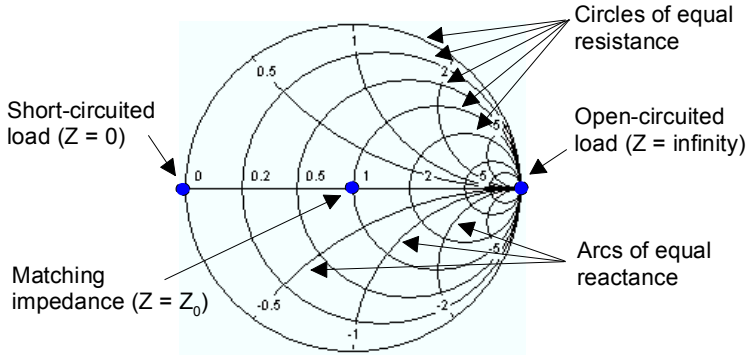
$$Z / Z_0 = (1 + \Gamma) / (1 - \Gamma)$$

From this equation it is easy to relate the real and imaginary components of the complex resistance to the real and imaginary parts of $\Gamma$

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

in order to deduce the following properties of the graphical representation in a Smith chart:

- Real reflection coefficients are mapped to real impedances (resistances).
- The center of the $\Gamma$ plane ($\Gamma = 0$) is mapped to the reference impedance $Z_0$, whereas the circle with $|\Gamma| = 1$ 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:

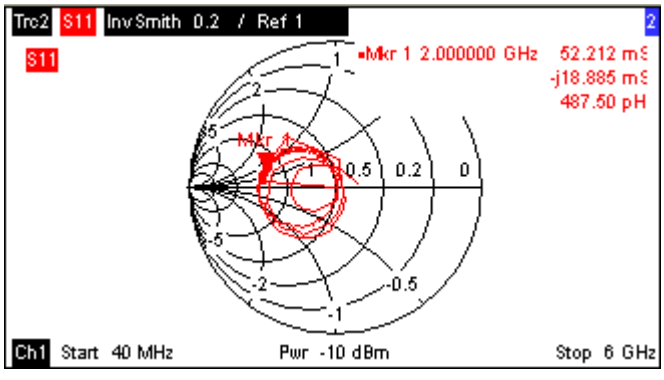
- The magnitude of the reflection coefficient of an open circuit (Z = infinity, I = 0) is one, its phase is zero.
- The magnitude of the reflection coefficient of a short circuit (Z = 0, U = 0) is one, its phase is $-180^0$.

## Inverted Smith Chart

The inverted Smith chart is a circular diagram that maps the complex reflection coefficients $S_{ii}$ to normalized admittance values. In contrast to the polar diagram, the scaling of the diagram is not linear. The grid lines correspond to points of constant conductance and susceptance.

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

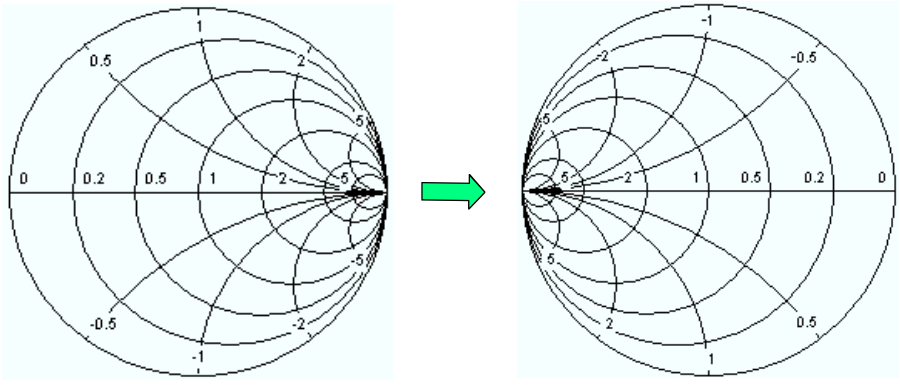
The following example shows an inverted Smith chart with a marker used to display the stimulus value, the complex admittance Y = G + j B and the equivalent inductance L (see marker format description in the help system).




TIP A comparison of the inverted Smith chart with the Smith chart and the polar diagram reveals many similarities between the different representations. In fact the shape of a trace does not change at all if the display format is switched from *Polar* to *Inverted Smith* or *Smith* – the analyzer simply replaces the underlying grid and the default marker format.

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 $\Gamma$ (e.g. $S_{11}$, $S_{22}$), then the unit 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):

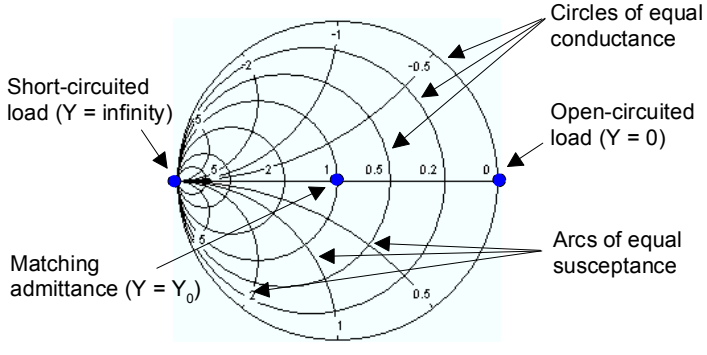
$$Y / Y_0 = (1 - \Gamma) / (1 + \Gamma)$$

From this equation it is easy to relate the real and imaginary components of the complex admittance to the real and imaginary parts of $\Gamma$

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

in order to deduce the following properties of the graphical representation in an inverted Smith chart:

- Real reflection coefficients are mapped to real admittances (conductances).
- The center of the $\Gamma$ plane ($\Gamma = 0$) is mapped to the reference admittance $Y_0$, whereas the circle with $|\Gamma| = 1$ 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 $-180^0$.
- The magnitude of the reflection coefficient of an open circuit (Y = 0, I = 0) is one, its phase is zero.

## Measured Quantities and Display 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 *Inverted Smith* lose their original meaning (standing wave ratio, normalized impedance or admittance) if they are used for transmission *S-parameters*, *Ratios* and other quantities.
- The complex *Impedances, Admittances, Z-parameters,* and *Y-parameters* are generally displayed in one of the Cartesian diagrams with linear vertical axis scale or in a polar diagram.
- The real *Stability Factors, DC Inputs,* and the *PAE* is generally displayed in a linear Cartesian diagram *(Lin Mag* or *Real).* 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-pParameters, impedances, admittances** | **Real quantities: Stability factors, DC input 1/2, PAE** |
|---|---|---|---|
| Lin Mag | ☑ | ☑ (default for Z-parameters, Y-parameters, impedances, admittances) | ☑ (default) |
| dB Mag | ☑ (default) | ☑ (default for wave quantities) | – |
| Phase | ☑ | ☑ | – |
| Real | ☑ | ☑ | ☑ |
| Imag | ☑ | ☑ | – |
| Unwrapped Phase | ☑ | ☑ | – |
| Smith | ☑ (reflection coefficients $S_{ii}$) | – | – |
| Polar | ☑ | – | – |
| Inverted Smith | ☑ (reflection coefficients $S_{ii}$) | – | – |
| SWR | ☑ (reflection coefficients $S_{ii}$) | – | – |
| Delay | ☑ (transmission coefficients $S_{ij}$) | – | – |

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

# Measured Quantities

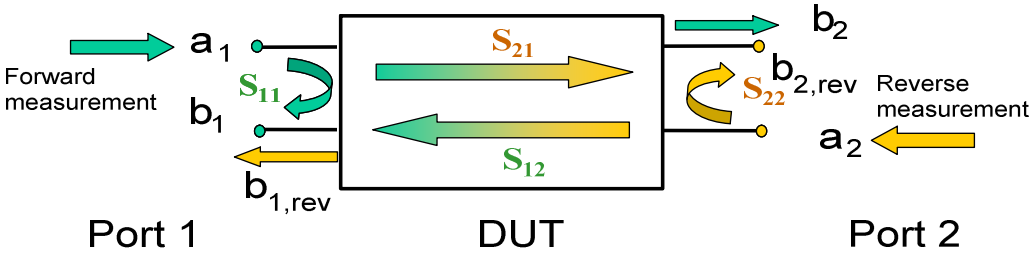
This section gives an overview of the measurement results of the network analyzer and the meaning of the different measured quantities. All quantities can be selected in the *Trace – Meas.* submenu.

NOTE

*The definitions in this and the following sections apply to general n-port DUTs. An analyzer with a smaller number of test ports provides a subset of the n-port quantities.*

## 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:




Extensions to the signal flow

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 so that an additional $a_2$ contribution occurs in forward measurements, 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:

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

The equation shows that the S-parameters are expressed as $S_{<out><in>}$, where <out> and <in> denote the output and input port numbers of the DUT.

Meaning of 2-port S-parameters

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

- $S_{11}$ isthe input reflection coefficient, defined as the ratio of the wave quantities $b_1/a_1$, measured at PORT 1 (forward measurement with matched output and $a_2 = 0$).
- $S_{21}$ is the forward transmission coefficient, defined as the ratio of the wave quantities $b_2/a_1$ (forward measurement with matched output and $a_2 = 0$).
- $S_{12}$ is the reverse transmission coefficient, defined as the ratio of the wave quantities $b_1$ (reverse measurement with matched input, $b_{1,rev}$ in the figure above and $a_1 = 0$) to $a_2$.

- $S_{22}$ is the output reflection coefficient, defined as the ratio of the wave quantities $b_2$ (reverse measurement with matched input, $b_{2,rev}$ in the figure above and $a_1 = 0$) to $a_2$, measured at PORT 2.

Meaning of squared amplitudes

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

| | |
|---|---|
| $\|a1\|^2$ | Available incident power at the input of a two-port (= the power provided by a generator with a source impedance equal to the reference impedance $Z_0$) |
| $\|a2\|^2$ | Available incident power at the output |
| $\|b1\|^2$ | Reflected power at the input of a two-port |
| $\|b2\|^2$ | Reflected power at the output |
| $10*\log\|S_{11}\|^2$ (= $20*\log\|S_{11}\|$) | Reflection loss at the input |
| $10*\log\|S_{22}\|^2$ | Reflection loss at the output |
| $10*\log\|S_{21}\|^2$ | Insertion loss at the input |
| $10*\log\|S_{12}\|^2$ | Insertion loss at the output |

### Multiport Parameters

The multiport S-parameters extend the standard 2-port S-parameters to a larger number of incoming 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 $S_{<out><in>}$.

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.

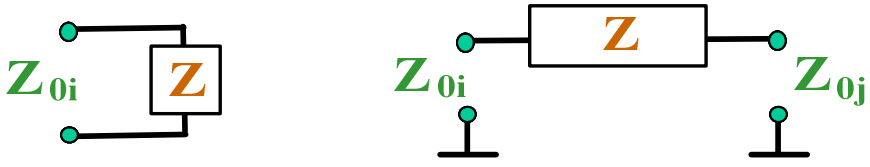
## Impedance Parameters

An impedance is the complex ratio between a voltage and a current. The analyzer provides two independent sets of impedance parameters, essentially based on different n-port circuit models:

- Matched-circuit impedances, converted from reflection S-parameters
- Open-circuit Z-parameters

### Converted Impedances

The converted, matched-circuit impedances describe the impedances of a DUT that is terminated at its outputs with the reference impedance $Z_{0i}$ displayed in the *Port Configuration* dialog. i numbers the analyzer/DUT port.




Reflection, $Z_{ii}$ Transmission, $Z_{ij}$

The analyzer converts the measured S-parameters to determine the matched-circuit impedances.

Relation with S-parameters

It is possible to express the matched-circuit impedances $Z_{ii}$ in terms of the reflection S-parameters $S_{ii}$ and vice versa. The reflection parameters are calculated according to:

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

where i numbers the analyzer/DUT port. The transmission parameters are calculated according to:

$$Z_{ij} = 2\cdot\frac{\sqrt{Z_{0i}\cdot Z_{0j}}}{S_{ij}} - \left(Z_{0i}+Z_{0j}\right), \quad i \neq j,$$

The two sets of reflection S-parameters and input/output impedances provide equivalent descriptions of the reflection in a linear n-port network. Another set of equivalent parameters is given by the matched-circuit admittances, defined as the inverse of the impedances.

**Examples:**

- $Z_{11}$ is the input impedance of a 2-port DUT that is terminated at its output with the reference impedance $Z_0$ (matched -circuit impedance measured in a forward reflection measurement).
- 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 $Z_0$. See *More Impedances* for detailed information.

TIP You can also read the converted impedances in a reflection coefficient measurement from the Smith chart.

## Z-Parameters

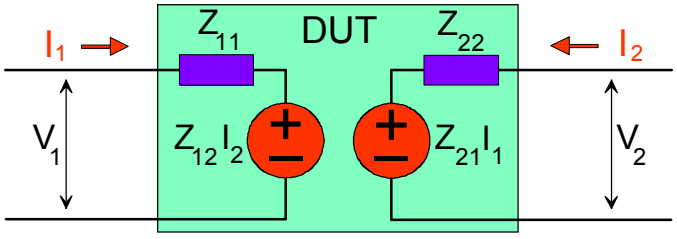
The Z-parameters describe the impedances of a DUT with open output ports (I = 0). The analyzer provides the full set of open-circuit impedances including the transfer impedances (i.e. the complete nxn Z-matrix for an n port DUT).

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

### 2-Port Open-Circuit Z-Parameters

In analogy to S-parameters, Z-parameters are expressed as $Z_{<out><in>}$, where <out> and <in> denote the output and input port numbers of the DUT.

The open-circuit 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 open-circuit 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.

## Admittance Parameters

An admittance is the complex ratio between a current and a voltage. The analyzer provides two independent sets of admittance parameters, essentially based on different n-port circuit models:

- Matched-circuit admittances, converted from S-parameters
- Short-circuit Y-parameters

### Converted Admittances

The converted (matched-circuit) admittances describe the admittances of a DUT that is terminated at its outputs with the reference impedance values $Z_{0i}$ displayed in the *Port Configuration* dialog. i numbers the analyzer/DUT port. The analyzer converts the measured S-parameters to determine the matched-circuit admittances. The converted admittances are the inverse of the converted impedances. The reflection parameters are calculated according to:

$$Y_{ii} = \frac{1}{Z_{0i}} \frac{1 - S_{ii}}{1 + S_{ii}} = 1 / Z_{ii} \ ,$$

where i numbers the analyzer/DUT port. The transmission parameters are calculated according to:

$$Y_{ij} = \frac{S_{ij}}{2 \cdot \sqrt{Z_{0i} \cdot Z_{0j}} - S_{ij} \cdot \left(Z_{0i} + Z_{0j}\right)} = 1 / Z_{ij}, \quad i \neq j, \quad i, j = 1, \ ..., \ 99$$

**Examples:**

- $Y_{11}$ is the input admittance of a 2-port DUT that is terminated at its output with the reference impedance $Z_0$ (matched -circuit admittance measured in a forward reflection measurement).
- The extension of the admittances to more ports and mixed mode measurements is analogous to S-parameters. $Y_{dd44}$ is the differential mode input admittance at port 4 of a DUT that is terminated at its other ports with the reference impedance $Z_0$. See *More Admittances* for detailed information.

You can also read the converted admittances in a reflection coefficient measurement from the inverted Smith chart.

### Y-Parameters

The Y-parameters describe the admittances of a DUT with output ports terminated in a short circuit (V = 0). The analyzer provides the full set of short-circuit admittances 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) in order to completely characterize a linear n-port network.

#### 2-Port Short-Circuit 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 open-circuit Z-parameters, the short-circuit 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.

## 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 a 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 amplitudes.

NOTE

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

## Wave Quantities

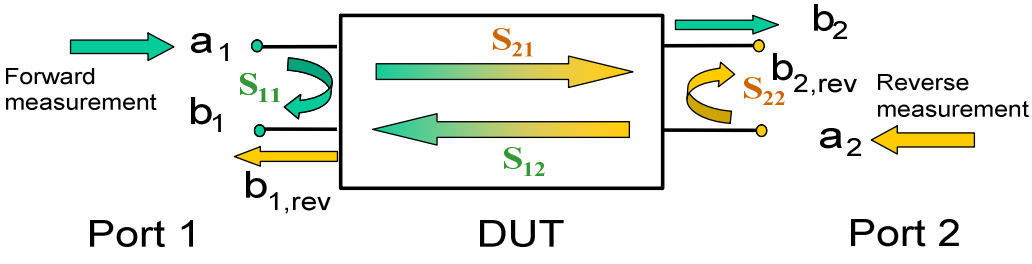
A wave quantity measurement provides the power of any of the transmitted or received waves. The power can be displayed in voltage (e.g. V or dBmV) or equivalent power units (e.g. W or dBm).

TIP 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 cancelled. Wave quantities are therefore suitable for the following measurement tasks:

- Analysis of non-linearities of the DUT.
- Use of the analyzer as a selective power meter.

The notation for wave quantities includes the direction and the test port number. Additionally, the source port must be specified. The letter a indicates a transmitted wave, b a received wave.




**Examples:**

- *a1 Src Port 1* is the outgoing wave at test port 1. In a standard S-parameter measurement, this wave is fed to the input port (port 1) of the DUT (forward measurement).
- *b1 Src Port 1* is the incoming wave at test port 1. In a standard S-parameter measurement, this is the reflected wave at port 1 of the DUT (forward measurement).

## 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 $b_i/a_j$ (ratio of the incoming wave to the outgoing wave at the test ports i and j of the DUT) are considered.

TIP 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 includes the two waves with their directions and test port numbers. Additionally, the source port must be specified. In analogy to wave quantities, the letter a indicates an outgoing wave, b an incoming wave.

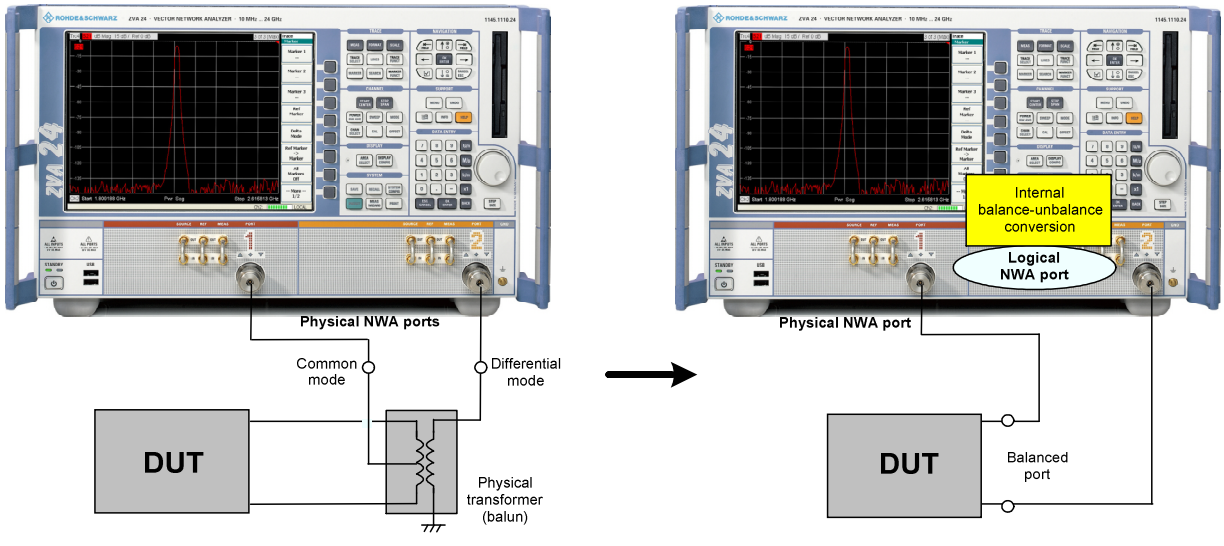
**Examples:**

- *b2/a1 Src Port 1* is the ratio of the incoming wave at port 2 and the outgoing wave at port 1 $b_2$ and $a_1$ and corresponds to the S-parameter $S_{21}$ (forward transmission coefficient).
- *b1/a1 Src Port 1* is the ratio of the wave quantities $b_1$ and $a_1$, measured at PORT 1, and corresponds to the S-parameter $S_{11}$ (input reflection coefficient).

## Balance - Unbalance Conversion

Balance-unbalance conversion is the simulation of one or more balance-unbalance transformers (baluns) integrated in the measurement circuit in order to convert the DUT ports from an unbalanced state into a balanced state and virtually separate 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



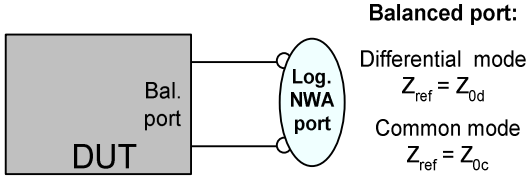
Balance-unbalance conversion avoids the disadvantages of real transformers:

- 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.

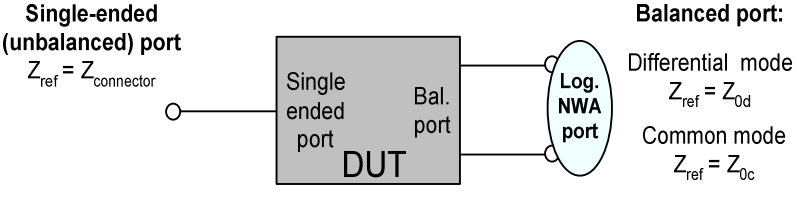
## Balanced Port Configurations

Defining a logical ports requires two physical analyzer ports. The ports of an analyzer are equivalent and can be freely combined. 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. The following types of balanced devices can be measured with 2-port, 3-port and 4-port analyzers:

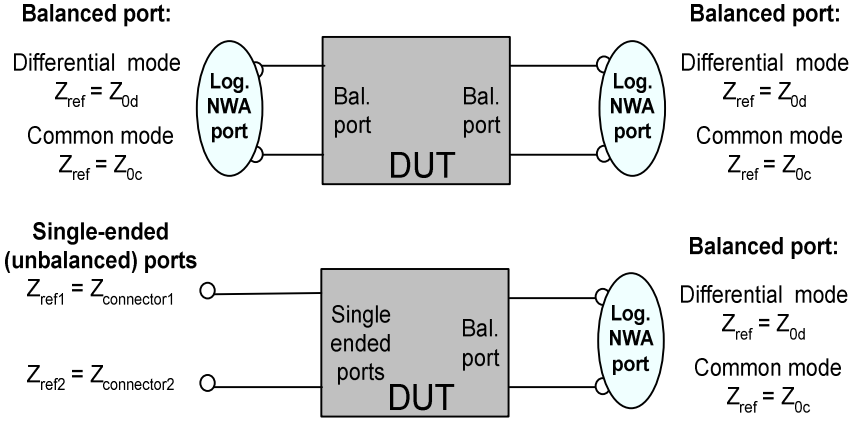
**2-port analyzers:** Reflection measurements on 1 balanced port




**3-port analyzers:** Reflection and transmission measurements on 1 balanced port




**4-port analyzers:** Reflection and transmission measurements on 1 or 2 balanced ports




A balanced port configuration is defined by simply selecting the pairs of physical ports that are combined to form balanced ports and defining the two reference impedances for the differential and common mode at each balanced port. All this is done in a single dialog; refer to the help system for details and measurement examples. The most commonly used balanced port configurations and impedances are predefined and can be selected in the *Measurement Wizard.*

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

## 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 as soon as a balanced port configuration is selected.

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

s: Singe-ended (for unbalanced ports)

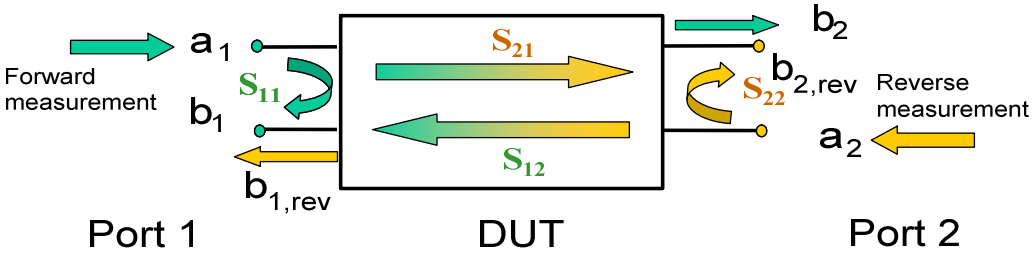
d: Differential mode (for balanced ports)

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:



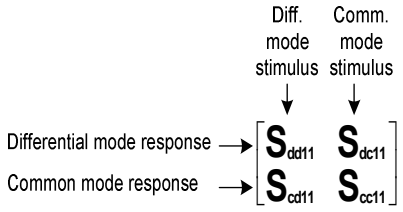

- $S_{<mout><min>11}$ is the mixed mode input reflection coefficient, defined as the ratio of the wave quantities $b_1$ (mode mout) to $a_1$ (mode min), measured at PORT 1 (forward measurement with matched output and $a_2 = 0$).
- $S_{<mout><min>21}$ is the mixed mode forward transmission coefficient,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

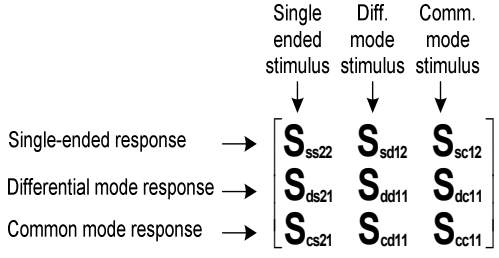
## Mixed Mode Parameters for Different Test Setups

The mixed mode parameter types available depend on the measured device and the port configuration of the analyzer. The following examples of mixed more parameters can all be obtained with a 4-port analyzer.

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.

# Calibration Overview

Calibration is the process of eliminating systematic, reproducible errors from the measurement results (system error correction). 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. The 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 the *Calibration Unit* (accessory ZV-Z51).

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.

In addition to the system error correction described in this section the analyzer supports source and receiver power calibrations using external power meters and generators. Refer to the help system for a comprehensive overview and comparison of the calibration types, including their use, properties and advantages.

## 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 by means of S-parameter tables stored in a file.

The analyzer provides a large number of predefined cal kits but can also import cal kit files and create new kits:

- A selection of predefined kits is available for all connector types. The parameters of these kits are displayed in the *Add/Modify 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 sub-dialogs.

Calibration kits and connector types are global resources; the parameters are stored independently and are available irrespective of the current setup.

## 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.

| Calibration Type | Standards | Parameters | Error Terms | General Accuracy | Application |
|---|---|---|---|---|---|
| Reflection Normalization | Open or Short | $S_{11}$ (or $S_{22}$, ...) | Reflection tracking | Low to medium | Reflection measurements on any port. |
| Transmission Normalization | Through | $S_{12}$, $S_{21}$ (or $S_{13}$,...) | Transmission tracking | Medium | Transmission measurements in any direction and between any combination of ports. |
| Full One-Port | Open, Short, Match[1)] | $S_{11}$ (or $S_{22}$, ...) | Reflection tracking, Source match Directivity, | High | Reflection measurements on any port. |
| One-Path Two-Port | Open, Short, Match[1)] (at source port), Through[2)] | $S_{11}$, $S_{21}$ (or $S_{22}$,...) | Reflection tracking, Source match, Directivity, Transmission tracking | Medium to high | Unidirectional transmission measurements in any direction and between any combination of ports. |
| TOSM (2-port, 3-port or 4-port) | Open, Short, Match[1)] (at each port), Through[2)] (between all combinations of 2 ports) | All | Reflection tracking, Source match, Directivity, Load match, Transmission tracking | High | Reflection and transmission measurements on DUTs with 2, 3 or 4 ports; classical 12-term error correction model. |
| TOM (2-port) | Open, Match (at both ports), Through | All | Reflection tracking, Source match, Directivity, Load match, Transmission tracking | High, implicit verification | Reflection and transmission measurements on DUTs with 2 ports. |
| TRM (2-port) | Reflect (equal at both ports), Match, Through | All | See TOM | High | Reflection and transmission measurements on DUTs with 2 ports, especially in test fixtures. |

| Calibration Type | Standards | Parameters | Error Terms | General Accuracy | Application |
|---|---|---|---|---|---|
| TRL (2-port) | Reflect (at both ports), Through, Line | All | See TOM | High, high directivity | Reflection and transmission measurements on DUTs with 2 ports, especially for planar circuits. Limited bandwidth. |
| TNA (2-port) | Through, Attenuation, Symmetric network | All | See TOM | High, lowest requirements on standards | Reflection and transmission measurements on DUTs with 2 ports, especially for planar circuits. |

1) Or any other 3 known one-port standards. 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. See remark above.

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

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.

## Full One-Port Calibration

A 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 full one-port calibration is more accurate than a normalization but is only applicable for reflection measurements.

## One-Path Two-Port Calibration

A one-path two-port calibration combines a full one-port calibration with a transmission normalization, so it 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 a second load port. 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 term.

A one-path two-port calibration requires only four standards to be connected (instead of 7 for a full two-port TOSM calibration) and 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 to be measured and the DUT is well matched, especially at the load port.

## TOSM Calibration

A TOSM (Through – Open – Short – Match) calibration requires the same standards as the one-path two-port 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 both source ports.
- The **transmission tracking** is determined for each direction.

TOSM calibration is provided for 2-port, 3-port or 4-port measurements. 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<br>+1 = 7 | 2 * 3<br>+2 * 1 = 8 | 2 * 3<br>+ 2 * 2 = 10 |
| 3 | 3 * 3<br>+ 2 + 1 = 12 | 3 * 3<br>+2 * (2 + 1) = 15 | 3 * 3<br>+ 2 * 2 * 3 = 21 |
| 4 | 4 * 3<br>+3 + 2 + 1 = 18 | 4 * 3<br>+2 * (3 + 2 + 1) = 24 | 4 * 3<br>+ 2 * 2 * 6 = 36 |

An open, through and match measurement is required at each port; in addition, a through must be measured between any combination of ports and in both directions. Therefore the number N of standard measurements for an n-port TOSM calibration is equal to

$$N = 3 \cdot n + 2 \cdot \binom{n}{2} = 3 \cdot n + 2 \cdot \frac{n!}{2! \cdot (n-2)!} = n \cdot (n+2).$$

The analyzer automatically performs each through measurement in both directions, so the number of connected standards is smaller than the number of measurements.

## TOM Calibration

A TOM (Through – Open – Match) calibration requires a low-reflection, low-loss through standard with an electrical length that may be different from zero, an open and a match. The characteristics of all standards must be fully known.

Implicit verification

The two one-port standards of the TOM calibration are connected to both ports. Together with the four S-parameters of the through standard, this results in 8 S-parameter measurements.

Despite the fact that TOM uses fewer standards than TOSM – the short is missing –, the 3 fully known standards provide more information than necessary. This is due to the system error model that TOM and the other TXX calibration procedures (see below) are based on. This model has only 7 error terms in contrast to the TOSM model, which contains 10 terms.

The redundancy of TOM is used for an implicit verification of the solution. Implicit verification helps to detect and avoid calibration errors, e.g. errors due to defective cables or loose connections. In the case of failure of that implicit verification a warning message appears. The user may then continue or cancel the calibration.

## TRM Calibration

A TRM (Through – Reflect – Match) calibration requires a low-reflection, low-loss through standard with an electrical length that may 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.

## TRL Calibration

A TRL (Through – Reflect – Line) calibration requires the two-port standards through and line, which are both assumed to be ideally matched. Beyond that, the through must be lossless. 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.

Frequency restrictions

The system of equations solved to derive the error terms is such that singularities occur whenever the length difference $\Delta L$ between the Through and the Line is equal to an integer multiple of half of the wave length:

$$\Delta L \neq n \frac{\lambda}{2}$$

As a rule, singularities are avoided with sufficient accuracy if the phase shift resulting from the length difference between through and 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.

TIP To shift the calibrated sweep range to smaller or larger frequencies, you can use a longer or shorter line.

**Example:** Suppose you want to perform a TRL calibration using coaxial air lines for the sweep range between 100 MHz and 4 GHz. The through standard has zero length. The ratio of the stop frequency to the start frequency is 40, therefore an accurate calibration with a single line standard is not possible. Proceed as follows:

1. Divide the sweep range into two subranges that meet the TRL calibration criterion, e.g. [100 MHz, 800 MHz] and [800 MHz, 4 GHz].

2. Determine the length $L_1$ and $L_2$ of the air lines from the following conditions in the center of the subranges:

$L_1 = \lambda$ (450 MHz)/4 = 16.7 cm; $L_2 = \lambda$ (2.4 GHz)/4 = 3.1 cm

3. Calibrate the subranges separately using the appropriate lines.

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 may be an alternative.

### 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 zero 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 may be an alternative.

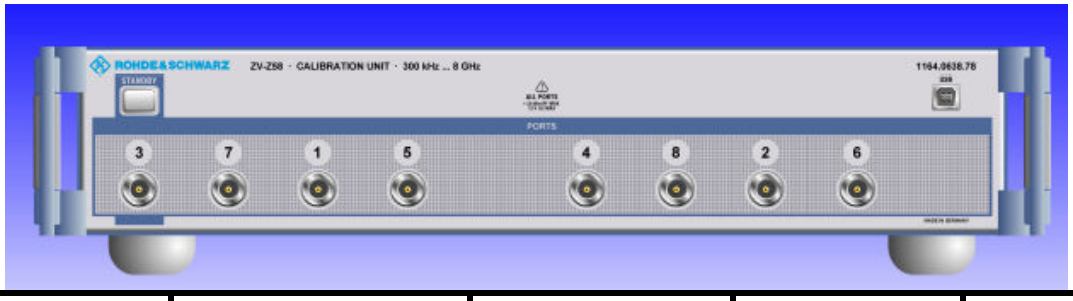
## Automatic Calibration

A *Calibration Unit* is an integrated solution for automatic system error calibration of vector network analyzers of the R&S ZVAB family. Rohde & Schwarz provides different types of calibration units:

- The 2- and 4-port calibration units R&S ZV-Z51, R&S ZV-Z52, and R&S ZV-Z53 are especially suited for R&S ZVB and R&S ZVA vector network analyzers.

| Calibration unit | Frequency range | Connector type | No. of ports | Order no. |
|---|---|---|---|---|
| R&S ZV-Z51 | 300 kHz to 8 GHz | type N | 4 | 1164.0515.70 |
| R&S ZV-Z51 | 300 kHz to 8 GHz | 3.5 mm | 4 | 1164.0515.30 |
| R&S ZV-Z52 | 10 MHz to 18 GHz | type N | 2 | 1164.0521.72 |
| R&S ZV-Z52 | 10 MHz to 24 GHz | 3.5 mm | 4 | 1164.0521.30 |
| R&S ZV-Z53 | 300 kHz to 18 GHz | type N | 2 | 1164.0473.72 |

- The 8- and 6-port calibration units R&S ZV-Z58 and R&S ZV-Z59 are especially suited for R&S ZVT vector network analyzers.




| Calibration unit | Frequency range | Connector type | No. of ports | Order no. |
|---|---|---|---|---|
| R&S ZV-Z58 | 300 kHz to 8 GHz | type N | 8 | 1164.0638.78 |
| R&S ZV-Z58 | 300 kHz to 8 GHz | 3.5 mm | 8 | 1164.0638.78 |
| R&S ZV-Z59 | 10 MHz to 20 GHz | 3.5 mm | 6 | 1164.0450.36 |

The units contain calibration standards that are electronically switched when a calibration is performed. The calibration kit data for the internal standards are 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 generally 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; see *TOSM Calibration.*
  - 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 don't wear out because they are switched electronically.

- **Limitations**

  Automatic calibration is less flexible than manual calibration: Some calibration types (TOM, TRM, TRL, TNA) are not available.

**Attention!**

Please observe the safety instructions on the technical information provided with the calibration unit to avoid any damage to the unit and the network analyzer. Safety-related aspects of the connection and operation of the units are also reported in the sections below.

## Connecting the Calibration Units to the Network Analyzer

The calibration units provide the following connectors:

- **Calibration units R&S ZV-Z51/-K52/-K53**
  - A USB type B connector at the rear is used to power-supply and control the unit. A USB cable for connection to the network analyzer is provided with the calibration unit.
  - Two or four RF connectors numbered 1 to 4 are to be connected to the test ports 1 to 4 of the analyzer. The connector type is either type N (f) or 3.5 mm (f) for all connectors.
- **Calibration units R&S ZV-Z58/-K59**
  - A USB type B connector at the front is used to control the unit. A USB cable for connection to the network analyzer is provided with the calibration unit.
  - The RF connectors numbered 1 to 8 / 6 are to be connected to the corresponding test ports of the analyzer. The connector type is type N (f) or 3.5 mm (f) for all connectors.

**To connect the unit...**

1. Switch on and start-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 on the front or rear panel of the analyzer. Connect the USB type B connector of the USB cable to the USB type B connector of the calibration unit. You can also connect the unit before switching on the analyzer.
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 “Performing an Automatic Calibration” below.

**Note:**

- The calibration unit is intended for direct connection to ZVA 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 cause damage to 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.

## Performing an Automatic Calibration

After connection and initialization, you can use the calibration unit as follows:

1. Connect ports n analyzer ports (n = 1 to 8, depending on your analyzer and calibration unit model and the number of ports to be calibrated) to n arbitrary ports of the calibration unit. Terminate all unused ports (no. n + 1 to 2, 4, or 8) with a 50 Ω match.
2. Perform the automatic calibration for the selected number of ports using the *Measurement Wizard* or the *Start Calibration* menu.
3. Remove the test cables from the unit, connect your DUT instead and perform calibrated measurements.

The assignment between the analyzer ports and the cal unit ports is detected automatically. If auto-detection fails (e.g. because of a high attenuation in the signal path), connect matching port numbers and click *Default Port Assignment* at the beginning of the calibration process.

***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 adaptors must be inserted between the calibration unit and the test ports.

- After connecting the unit to the USB port, allow for a sufficient warm-up time (see *"Specifications"*) before starting the calibration.
- To ensure best accuracy the analyzer automatically reduces the source power to –10 dBm. If the test setup contains a large attenuation, deactivate *Automatic Power Reduction for Calibration Unit* in the initial tab of the *System Config* dialog and ensure an input power of –10 dBm at the ports of the calibration unit (please also refer to the separate *"Specifications"* document).

**Attention!**

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.

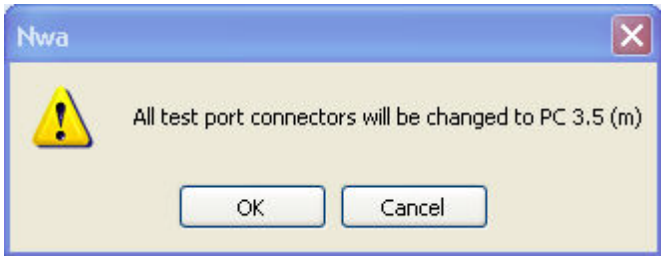
The calibration type depends on the number of ports. If a single port is calibrated, the analyzer uses full one-port calibration. For 2 and more ports, the analyzer provides the following options:

- Full n-port calibration for n calibrated ports.
- n full one-port calibrations for n calibrated ports.

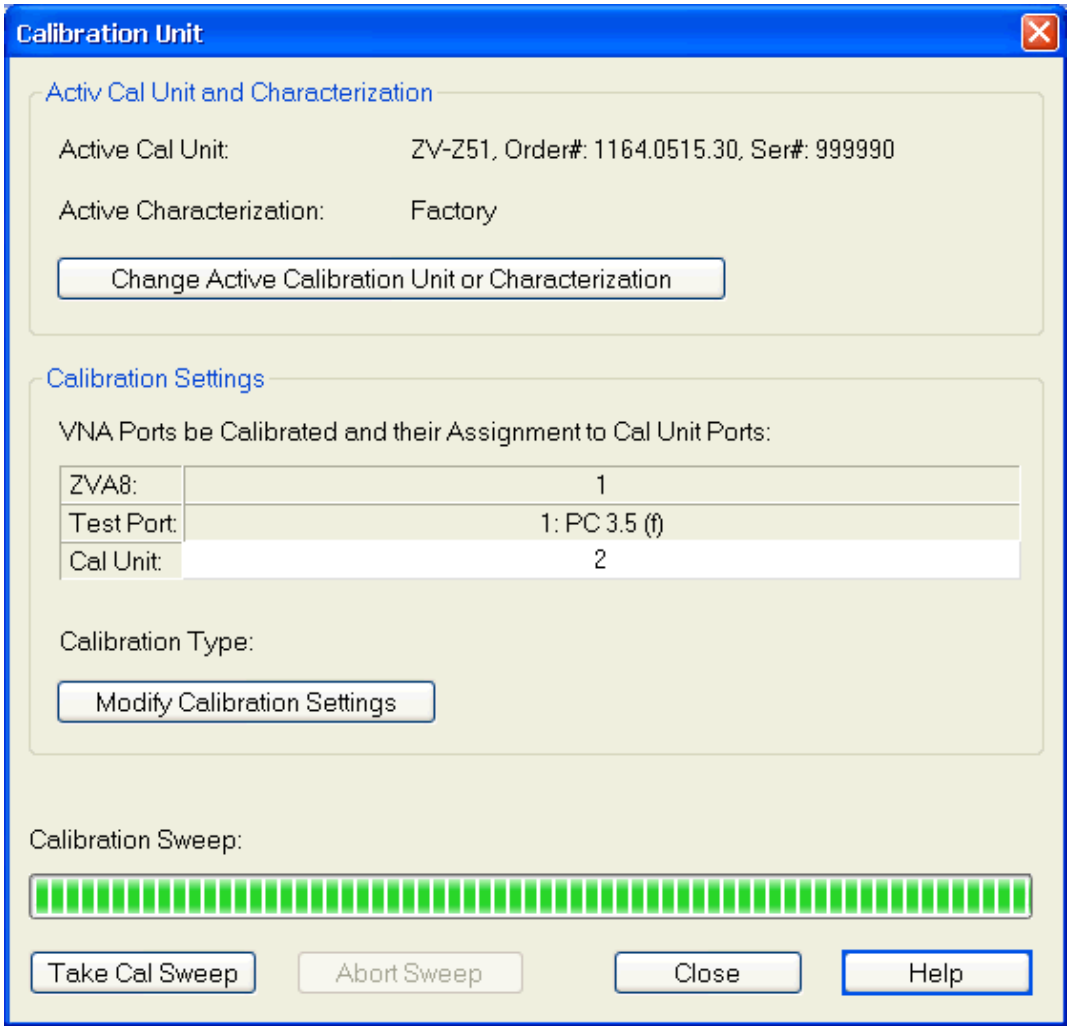
**Remote control:**
```
ENSe<Ch>:]CORRection:COLLect:AUTO
[SENSe<Ch>:]CORRection:COLLect:AUTO:PORTs
[SENSe<Ch>:]CORRection:COLLect:AUTO:PORTs:CONNection?
```

## Calibration Procedure

The automatic calibration is started on clicking *Calibration Unit* in the *Start Calibration* menu. When the test port connectors in the *Port Configuration* dialog do not fit to the connectors of the calibration unit, a warning appears:

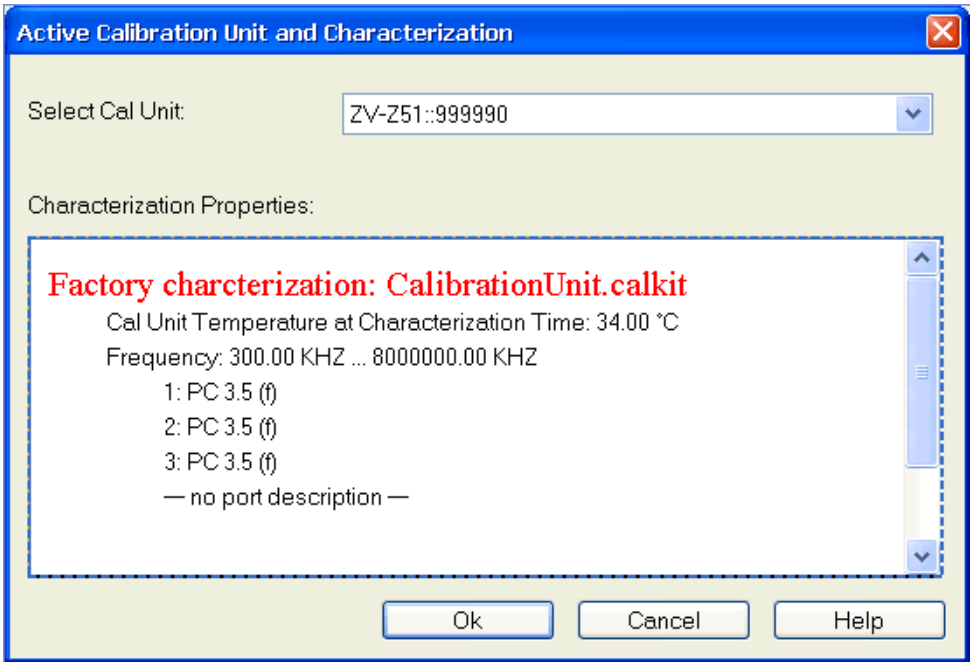



*OK* changes the connector type assignment of the test ports and starts the calibration. The *Calibration Unit* dialog shows the active cal unit type and characterization, the calibrated ports, and the progress of the calibration procedure.

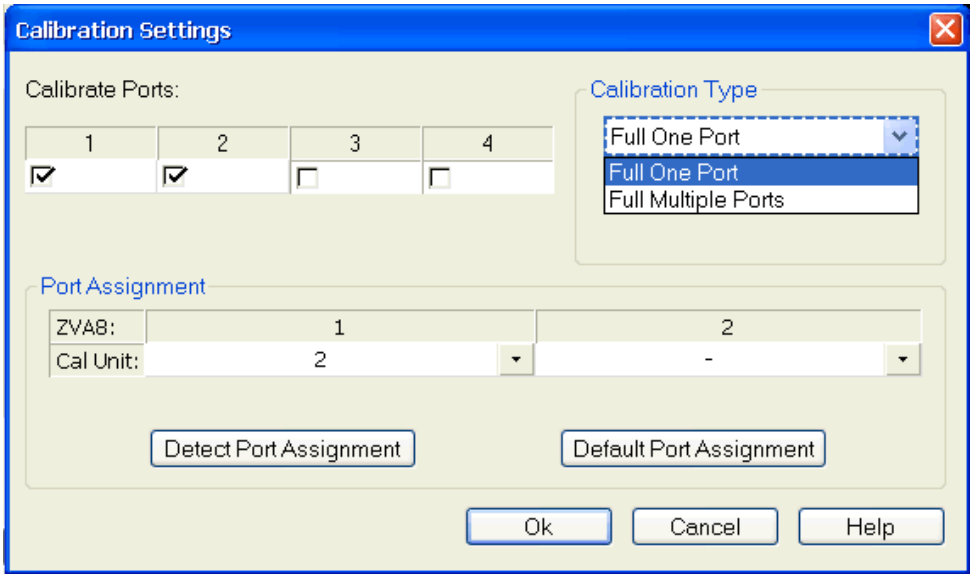



In the dialog, you can:

- Click ***Change Active Calibration Unit or Characterization*** in case that several cal units are connected to the analyzer and you wish to select another unit or characterization.




- Click ***Modify Calibration Settings*** to change the calibration type, the calibrated ports and the port assignment.




In the *Calibration Settings* dialog, you can:

- Selectthe network analyzer ports to be calibrated (*Calibrate Ports).*
- Select the *Calibration Type.* Either a *Full One Port* calibration at each port or a *Full Multiple Ports* calibration for all ports.
- Detectthe current port assignment between the analyzer and the connected calibration unit in order to overwrite the *Port Assignment* table.
- Restore the *Default Port Assignment* (connect port no. n of the analyzer to port no. n of the cal unit). This is important if the automatic detection of the port assignment fails.

- Click *Take Cal Sweep* to start the necessary calibration sweeps. The calibration is started automatically if automatic port detection is successful and the cal unit temperature is within the validity range.
- *Abort* an ongoing calibration sweep.

The *Calibration Unit* dialog is closed automatically when all standards have been measured, the system error correction data has been calculated, and the result is applied to the active channel. The network analyzer is ready to perform calibrated measurements.

# Contents of Chapter 4


**4 GUI Reference ..... 4.1**

**Control Menus ..... 4.2**
Restore ..... 4.2
Move ..... 4.2
Size ..... 4.2
Minimize ..... 4.3
Maximize ..... 4.3
Close ..... 4.3
Next ..... 4.3

**File Menu ..... 4.4**
New ..... 4.5
Close ..... 4.5
Print ..... 4.5
Print Dialog ..... 4.5
Print Preview ..... 4.6
Page Setup ..... 4.7
Page Setup Dialog ..... 4.7
Print to Clipboard ..... 4.9
Print to File ..... 4.9
Print to File Dialog ..... 4.9
Save ..... 4.11
Save As Dialog ..... 4.11
Open / Recall ..... 4.12
Open File Dialog ..... 4.12
Recent File ..... 4.12
Exit ..... 4.13

**Trace Menu ..... 4.14**
Measure ..... 4.15
S11, S12, S21, S22 ..... 4.16
More S-Parameters ..... 4.17
Single-Ended and Mixed Mode Parameters ..... 4.18
Ratios ..... 4.18
Impedance ..... 4.25
Admittance ..... 4.28
Z-Parameters... ..... 4.30
Y-Parameters ..... 4.31
Stability Factor... ..... 4.33
Imbalance ..... 4.34
All S-Params ..... 4.35
DC Inputs ..... 4.36
Power Sensor ..... 4.40
Format ..... 4.41
dB Mag ..... 4.43
Phase ..... 4.43

Smith ....4.43
Polar ....4.44
Delay ....4.44
Aperture ....4.45
SWR ....4.46
Lin Mag ....4.47
Real ....4.47
Imag ....4.48
Inverted Smith ....4.48
Unwrapped Phase ....4.48
Scale ....4.49
Autoscale ....4.50
Autoscale All ....4.50
Scale Div. ....4.50
Ref. Value ....4.50
Ref. Position ....4.51
Max and Min ....4.51
Ref. Val = Marker ....4.51
Trace Select ....4.52
Next Trace ....4.53
Select Trace ....4.53
Add Trace ....4.54
Add Trace + Diag. Area ....4.54
Delete Trace ....4.54
Assign Channel ....4.54
Assign Diag. Area ....4.55
Trace Manager ....4.55
Trace Funct ....4.58
Data -> Mem ....4.60
Math = Data/Mem ....4.60
Show Data ....4.61
Show Mem ....4.61
More Mem ....4.61
-> Mem Dialog ....4.62
All Data -> Mem ....4.63
Math = User Def ....4.64
User Def. Math... ....4.64
Transform ....4.67
Trace Statistics ....4.80
Smoothing On ....4.86
Smoothing Aperture ....4.86
Import/Export Data ....4.87
Shift Stimulus Value... ....4.94
Shift Response Value ....4.95
Max Hold On ....4.95
Restart Hold ....4.96
Marker ....4.96
Marker 1/2/3 ....4.98
Ref. Marker ....4.98
Delta Mode ....4.99
Ref. Marker -> Marker ....4.99
All Markers Off ....4.99

Coupled Markers ..... 4.99
Discrete Marker ..... 4.100
Marker Format ..... 4.100
More Markers ..... 4.102
Marker Properties ..... 4.102
Export Markers ..... 4.103
Search ..... 4.103
Max Search ..... 4.104
Min Search ..... 4.105
Next Peak ..... 4.105
Peak Search > ..... 4.105
< Peak Search ..... 4.105
Target ..... 4.106
Define Peak ..... 4.108
Search Range ..... 4.108
Marker Tracking ..... 4.110
Bandfilter ..... 4.110
Marker Funct ..... 4.116
Center = Marker ..... 4.117
Start = Marker ..... 4.117
Stop = Marker ..... 4.117
Span = Marker ..... 4.117
Ref. Value = Marker ..... 4.117
Max = Marker ..... 4.118
Min = Marker ..... 4.118
Zero Delay at Marker ..... 4.118
Trace - Lines ..... 4.118
Show Limit Line ..... 4.119
Limit Check On ..... 4.120
Fail Beep On ..... 4.121
Define Limit Line ..... 4.121
Horizontal Line ..... 4.127
Global Limit Check ..... 4.127

**Channel Menu ..... 4.129**
Stimulus ..... 4.129
Stimulus and Sweep Types ..... 4.131
CW Frequency ..... 4.132
Power Bandwidth Average ..... 4.132
Power ..... 4.133
Step Attenuators (R&S ZVA and R&S ZVB Analyzers) ..... 4.133
Step Atten. b2 (R&S ZVA Analyzers) ..... 4.136
RF Off ..... 4.136
Meas. Bandwidth ..... 4.136
Average Factor ..... 4.138
Sweep ..... 4.138
Sweep Type ..... 4.139
Pulsed Measurements (for R&S ZVA and R&S ZVT) ..... 4.150
Number of Points ..... 4.157
Frequency Step Size ..... 4.158
Sweep Time ..... 4.158
Trigger ..... 4.159

Single (All Chans) ....4.163
Restart ....4.163
Define Restart ....4.164
Mode ....4.164
Port Configuration ....4.165
True Differential Mode (for R&S ZVA, all models, and R&S ZVT 8) ....4.183
Harmonics ....4.191
Mixer ....4.197
Virtual Transform ....4.207
Alternate ....4.218
Spurious Avoidance ....4.219
ALC (All Chans) ....4.220
Low Phase Noise ....4.221
Channel Select ....4.221
Next Channel ....4.222
Select Channel ....4.222
Add Channel + Trace ....4.223
Add Channel + Trace + Diag. Area ....4.223
Delete Channel ....4.224
Channel Manager ....4.224
Calibration ....4.225
Start Cal ....4.226
Guided Calibration ....4.229
Repeat Previous Cal ....4.240
Recall Last Cal Set ....4.240
Correction Off ....4.240
Power Calibration ....4.240
Status Messages for Power Calibration ....4.247
Interpolation and Extrapolation ....4.248
Power Correction On ....4.248
Power Correction Off ....4.248
Calibration Manager ....4.249
Calibration Kits ....4.250
Offset ....4.263
Electrical Length ....4.265
Mechanical Length ....4.266
Delay ....4.267
Auto Length ....4.267
Auto Length and Loss ....4.268

**Display Menu ....4.270**
Display - Area Select ....4.270
Next Diag. Area ....4.271
Select Diag. Area ....4.272
Add Diag. Area + Trace ....4.272
Delete Diag Area ....4.272
Overlay all ....4.273
Split all ....4.273
Maximize ....4.274
Dual Split ....4.275
Triple Split ....4.275
Quad Split ....4.276

Split Manager ....4.277
Title ....4.279
Display Config ....4.279
Softkey Bar ....4.280
Hardkey Bar ....4.280
Title Bar ....4.281
Status Bar ....4.281
Color Scheme ....4.281
Define User Color Scheme ....4.282
Frequency Info ....4.284

**System Menu ....4.285**
System Config ....4.285
External Power Meters ....4.288
Add/Modify External Power Meter ....4.291
External Generators ....4.291
Add/Modify External Generator ....4.294
Int. Reference ....4.295
Ext. Reference ....4.295
GPIB Address ....4.296
Service Function ....4.296
Preset ....4.296
Undo ....4.296
Redo ....4.297
External Tools ....4.297
System Config ....4.297
Converter Control (Option R&S ZVA-K8, for R&S ZVA and R&S ZVT Network Analyzers) 4.297
Test Setup and Principle of the Measurement ....4.298
Frequency Converter ....4.298
Connecting the Frequency Converters ....4.300
Calibration ....4.302
Measurement ....4.303
LXI Configuration ....4.304
LXI Configuration ....4.304
LXI Browser Interface ....4.305
LAN Configuration ....4.307
Ping ....4.308
Measurement Wizard ....4.309

**Info Menu ....4.313**

**Window Menu ....4.314**
New ....4.314
Close ....4.315
Cascade ....4.315
Tile ....4.315
Maximize ....4.315
1 Set<n>.zvx ....4.315

**Help Menu ....4.316**

Help Topics.....4.316
About Nwa... .....4.316

# 4 GUI Reference

This chapter explains in detail all functions of the analyzer and their application. It is organized according to the menus/softkey groups of the user interface:

All topics in this chapter can be called up directly using the HELP key in the menus or the *Help* buttons in the dialogs. A link at the end of each function description leads to the corresponding remote control command.

For a general overview of the analyzer's capabilities and their use refer to the *System Overview*.

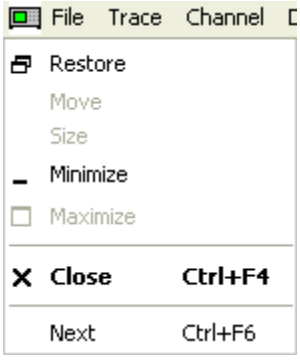
# Control Menus

The *Control* menus provide standard Windows™ functions to control windows. The analyzer provides two types of *Control* menus with analogous function:

- Clicking the icon opens the *Control* menu for the main application window. To access this icon the *Title Bar* of the main application window must be open.
- Clicking a icon opens the *Control* menu for an individual setup window. If a setup window is maximized, the icon is placed to the left of the *File* menu.

Double-clicking a control icon is the same as clicking the icon or the *Close* menu command.

(No direct access via front panel keys)




The *Control* menu contains the following functions:

- *Restore* returns the active window to its size and position.
- *Move* displays a cursor symbol to moves the active window within the available space.
- *Size* displays a cursor symbol to size the active window.
- *Minimize* reduces the active window to an icon.
- *Maximize* enlarges the active window to fill the available space.
- *Close* closes the application.
- *Next* switches to the next open setup window.

## Restore

Returns the maximized or minimized active window to its size and position. *Restore* is available after a *Maximize* or *Minimize* command only.

## Move

Displays a four-headed arrow to move the active window with the arrow keys.

NOTE This command is unavailable for maximized windows.

## Size

Returns the maximized or minimized active window to its size and position. *Restore* is available after a

*Maximize* or *Minimize* command only.

This command is unavailable for maximized windows.

## Minimize

Reduces the active window to an icon.

## Maximize

Enlarges the active window to fill the available space.

## Close

Closes the active setup window (setup control menu) or ends the analyzer session (application control menu).

- In a setup window the analyzer suggests to save changes to the setup before closing it. If a setup is closed without saving, all changes made since the last time it was saved are lost.
- In the application window the analyzer prompts you to save documents with unsaved changes.

The *Close* application command is equivalent to the *Exit* command in the *File* menu. Moreover *Close* is the same as double-clicking a *Control* menu icon or clicking the icon in the title bar of the active window.

## Next

Selects the next diagram area as the active diagram area. This command is available in setup control menus only and is disabled if only one setup is defined.

# File Menu

The *File* menu provides standard Windows™ functions to create, save, recall or print setups, to copy the current screen and to shut down the application.

**Setups**

A setup comprises a set of diagram areas with all displayed information that can be stored to a NWA setup file (*.zvx). Each setup is displayed in an independent window.

- The Window menu commands arrange different windows on the screen.
- The *File* menu is used to organize setups.

The *File* menu contains the following functions:

- *New* creates a new setup and opens a new setup window.
- *Close* closes an opened setup window.
- *Print* prints a setup.
- *Print Preview* displays the setup on the screen as it would appear printed.
- *Page Setup...* selects a printer and printer connection.
- *Print to Clipboard* copies the screen contents to the clipboard in order to be pasted into another application.
- *Print to File...* opens a dialog to select a file and file format to store the screen contents.
- *Save* saves an opened setup to a specific file.
- *Open/Recall* recalls an existing setup from a file.
- *Window* displays the softkeys of the *Window* menu.
- *1 Set<n>.zvx* etc. is a list of the last 4 setups stored in the current or in previous sessions.
- *Exit* closes the application.

## New

Creates a new setup and opens a new setup window. The new setup is named *Setup<n>* where <n> is the current number for all created setups.

To open an existing setup, select File – Open / Recall. To rename a setup, use *File – Save As*.

**Remote control:** `MEMory:DEFine "<setup_name>"`

## Close

Closes an opened setup window. The analyzer suggests to save changes to the setup before closing it. If a setup is closed without saving, all changes made since the last time it was saved are lost.

**Remote control:** `MEMory:DELete[:NAME] "<setup_name>"`

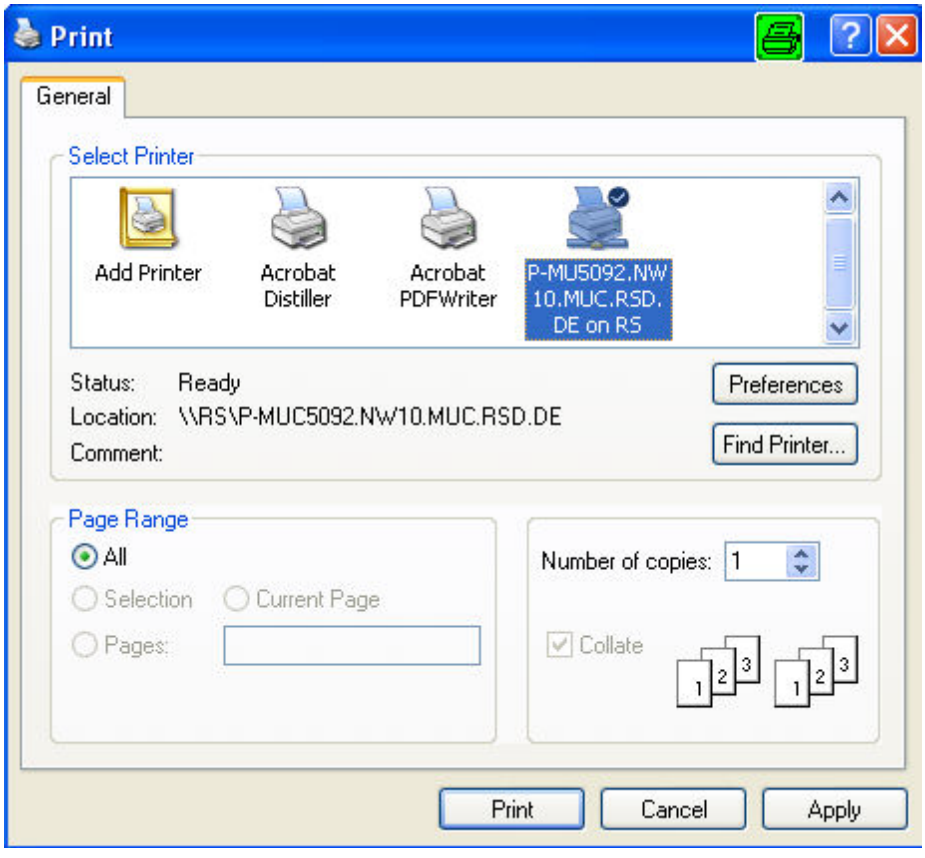
## Print

Prints the active setup. This command opens a standard Windows™ *Print* dialog box to, specify the range of pages to be printed, the number of copies, the destination printer, and other printer setup options.

**Remote control:** The commands in the `HCOPy...` system provide the printer settings. `HCOPY[:IMMediate]` initiates the print.

### Print Dialog

Provides options to specify how the setup should be printed. *Print* is opened by means of the *File – Print...* command.




The following printer options are remembered when the dialog is closed:

- *Select Printer* displays the active printer and printer connection. The printers and printer connections can be selected from the *Name* drop-down list.
- *Preferences* opens a dialog to select the layout and paper format and specify further printer options.
- *Find Printer* opens a dialog to select and install a printer that is not yet installed on the instrument.
- *Page Range* specifies the pages to be printed:

***All*** prints the entire setup.

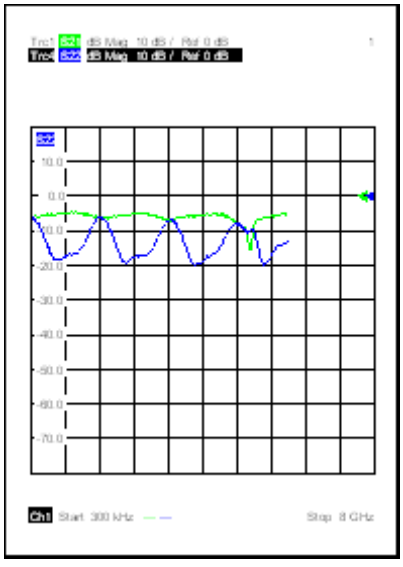
- *Number of Copies* specifies the number of copies to be printed for the above page range.

To select further printer options, open the *Print Setup* dialog.

**Remote control:** `HCOPy:DESTination`
`HCOPY[:IMMediate].`

## Print Preview

Displays the active setup as it would appear when printed. The analyzer generates one printed page for each diagram area in the setup. The main window is replaced with a print preview window in which one or two pages/diagram areas are displayed in their printed format.



Across the top of the screen the print preview toolbar offers options to view either one or two pages/diagram areas at a time; move back and forth through the setup; zoom in and out of pages/diagram areas (2 steps possible); initiate a print job or close the preview without printing:

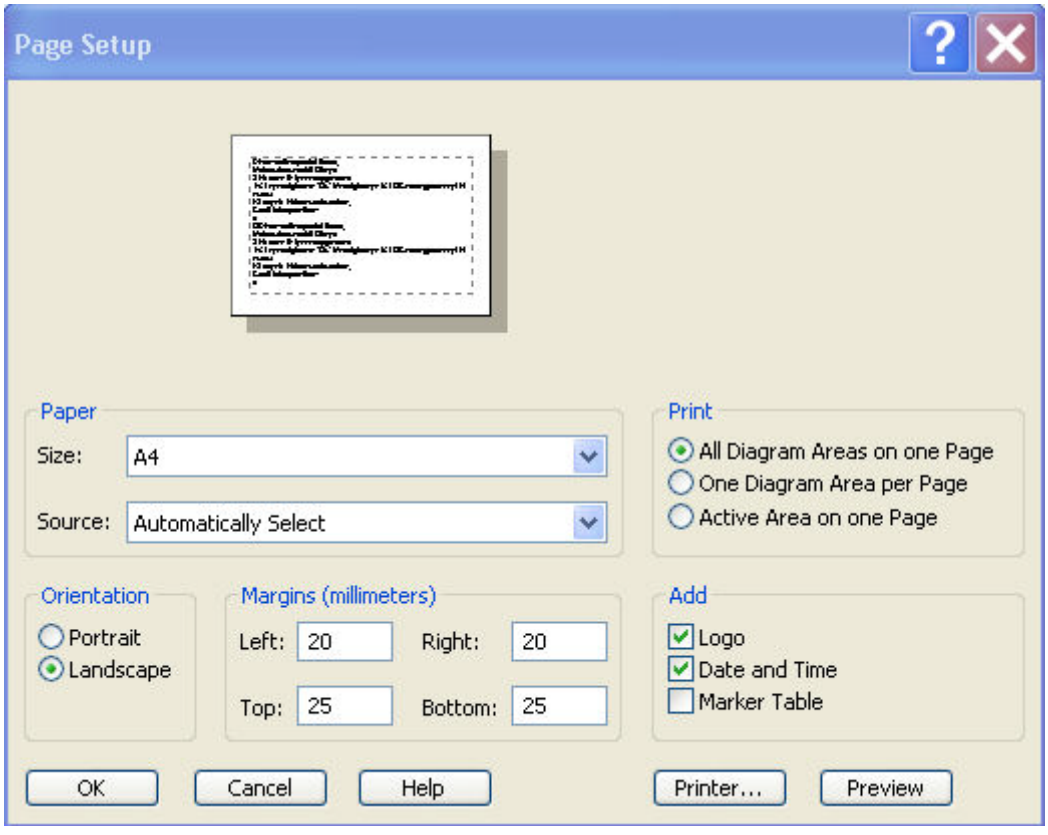
**Remote control:** No commands for check on screen.
`HCOPY[:IMMediate]` initiates the print.

## Page Setup

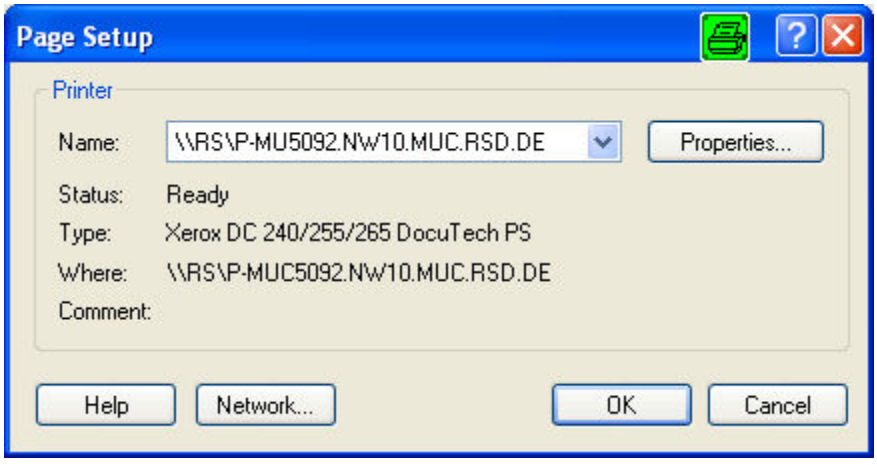
Selects a printer and a printer connection. This command presents a Page Setup dialog box to specify the printer and its connection.

### Page Setup Dialog

Provides options to specify how the document should be printed. *Page Setup* is opened by means of the *File – Page Setup...* command.




- *Paper – Size* specifies the size of paper that the document is to be printed on.
- *Paper – Source* specifies the paper source, because some printers offer multiple trays for different paper sources.
- The radio buttons in the *Print* panel specify whether or not different diagram areas are printed on different pages.
- The radio buttons in the *Orientation* panel specify whether the pages are printed in Portrait or Landscape format.
- The *Margins* panel contains input fields to define a margin on each of the four edges of the pages.
- The check boxes in the *Add* panel select additional information to be included in the printed document.
- *Printer...* opens the *Page Setup* dialog to select the printer and printer connection. The printers can be selected from the *Name* drop-down list. *Network...* opens a dialog to configure the printer connections. *Properties* opens a dialog to select the layout and paper format and specify further printer options.




- *Preview* shows the active setup as it would appear when printed.

To select the *Print Range* and start printing, open the *Print* dialog. To install printers and configure ports use the *Control Panel*.

**Remote control:**
```
HCOPy:DESTination
HCOPy:DEVice:LANGuage
HCOPy:ITEM:ALL
HCOPy:ITEM...
HCOPy:PAGE...
HCOPy:ORIentation
HCOPy:WINDow
HCOPY[:IMMediate].
```

## Print to Clipboard

Copies the current screen or a portion of it to the clipboard in order to be pasted into another application.

Use *Print to File* to copy the screen contents to a file with a selectable bitmap format. *Print to File* also provides options to specify which portion of the screen is copied to the clipboard.

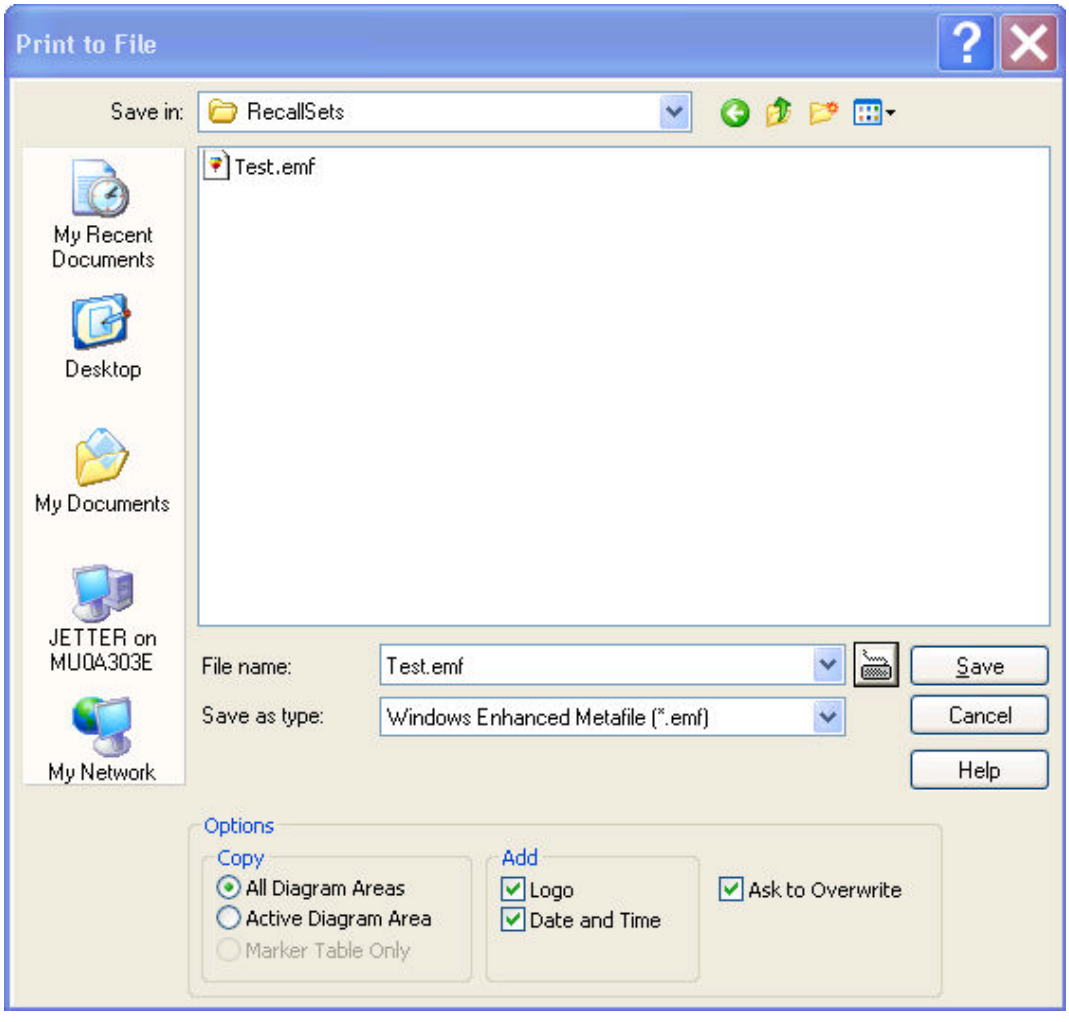
## Print to File

Opens a *Print to File* dialog to select a file and file format to store the current screen contents.

**Remote control:**
```
HCOPy:DESTination 'MMEM'
```

## Print to File Dialog

Provides options to specify how the screen contents are stored to a file. *Copy Display* is opened by means of the *File – Copy Screen to File* command.




- *Save in* specifies the drive and directory in which the screen contents are 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 *What's this?* help).
- *File Name* specifies a file name to save the screen contents. The analyzer adds the extension (e.g. *.emf) selected in the *Save As Type* box.
- *Save* saves the screen contents in the selected file and directory and closes the dialog. The print options are maintained and can be reused when *Print to File* is used for the next time.
- *Cancel* closes the dialog without saving the screen contents.
- The *Options...* panel provides options to specify whether the entire screen *(All Diag. Areas)* or a portion of it *(Active Diag. Area, Marker Table Only)* is copied, whether or not the logo The default R&S logo (file Logo.gif) is stored in the Resources\Images subdirectory of the NWA program directory and can be replaced by another logo. , date and time is placed above the selected output, and whether the analyzer displays a message to be confirmed before overwriting an existing file. The options are remembered when the dialog is closed.

NOTE After a file has been saved, the selected copy *Options* are also used if the screen is copied to the clipboard.

**Remote control:**
```
HCOPy:DESTination 'MMEM'
HCOPy:DEVice:LANGuage
HCOPy:ITEM:ALL
HCOPy:ITEM...
```

```
HCOPy:PAGE...
HCOPy:WINDow
HCOPY[:IMMediate].
```

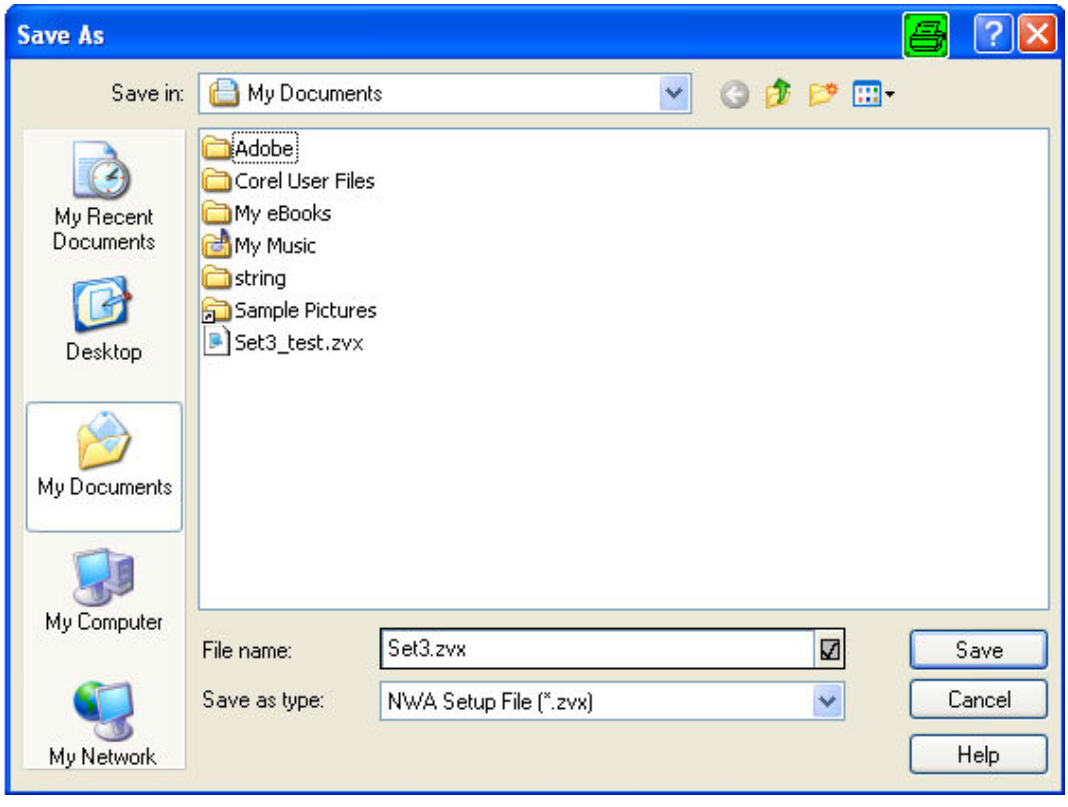
## Save

Saves and names the active setup. The analyzer opens a standard Windows™ *Save As* dialog box to select a NWA setup file name (*.zvx) and location for the setup file.

**Remote control:** `MMEMory:STORe:STATe 1,"<file_name>"`

### Save As Dialog

Specifies the name and location of a particular file (e.g. a NWA setup file) to save:




- *File Name* specifies a file name to save the current data (e.g. the setup). The analyzer adds the extension (e.g. *.zvx) in the *Save As Type* box.
- *Save 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 *What's this?* help).
- *Save* saves the data (e.g. the active setup) in the selected file and directory and closes the dialog.
- *Cancel* closes the dialog without saving the data.

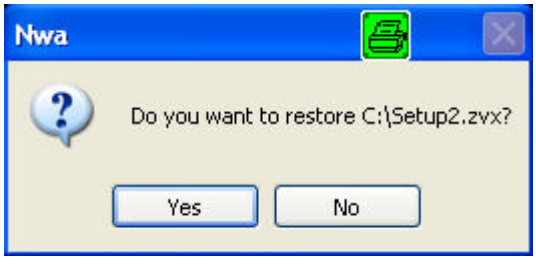
*The Save As dialog is used to store various data types (e.g. cal kit data, limit lines, sweep*

*segment lists, ...). Depending on its use the dialog is opened with different file locations and data types. File locations (directories) are remembered when the dialog is closed. To restore default directories use the System – System Configuration – Resets dialog.*

## Open / Recall

Recalls an existing setup from a file. The analyzer opens a standard Windows™ *Open File* dialog box to select the file from all NWA setup files (*.zvx) stored on the file system.

It is not possible to open the same setup in different windows. If an opened setup is changed and an attempt is made to open the same setup again the analyzer displays a warning:




*Yes* overwrites the changes in the opened setup, *No* closes the message box, leaving the opened setup unchanged.

It is possible to create and open several setup files with the same contents but different names or locations. To open/recall a setup file (*.zvx) you can also use the Windows Explorer and simply double-click the file or drag and drop the file into the NWA application. The imported setup becomes the active setup.

**Remote control:** `MMEMory:LOAD:STATe 1,"<file_name>"`

### Open File Dialog

Specifies the name and location of a particular file (e.g. a NWA setup file) to open:

- *Look in* specifies the drive and directory in which the file to open 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 *What's this?* help).
- *File Name* specifies a file name (e.g. a setup file, *.zvx) to open. The file can be selected by clicking on the directory overview above.
- *Open* opens selected file and closes the dialog.
- *Cancel* closes the dialog without opening a setup file.

## Recent File

Is a placeholder which is replaced by a list of the last 4 setups stored in the current or in previous sessions. Clicking an item in the list opens the corresponding setup.

Use the setup list in the *Window* menu to switch between different open setups.

**Remote control:** `MEMory:CAT?` (returns a list of the loaded files).

## Exit

Ends the analyzer session. The analyzer prompts you to save documents with unsaved changes.

This command is equivalent to the *Close* command on the application Control menu.

# Trace Menu

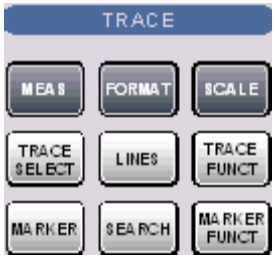
The *Trace* menu provides all trace settings and the functions to select, modify and store different traces. In addition the menu provides the marker, search and limit check functions.

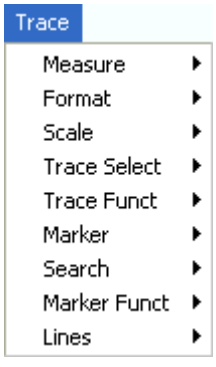
**Traces**

A trace is a set of data points that can be displayed together in a diagram area. The trace settings specify the mathematical operations used in order to obtain traces from the collected 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 complement the definitions of the *Channel* menu. Each trace is assigned to a channel; see *Traces, Channels and Diagram Areas*. The channel settings apply to all traces assigned to the channel.



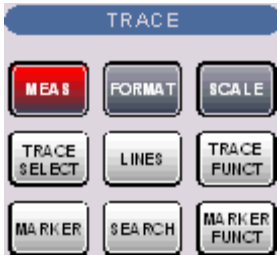



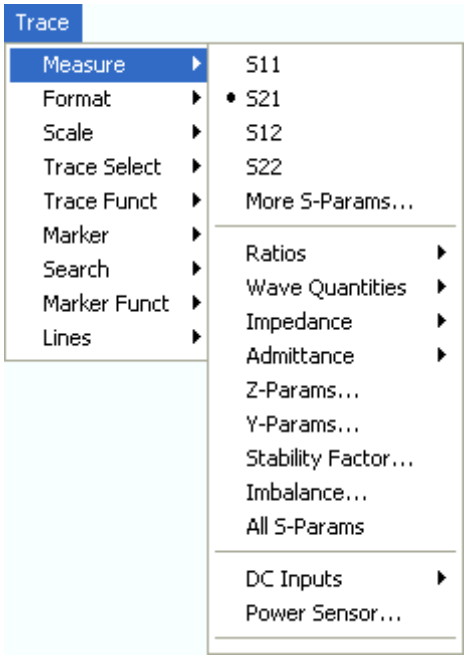

The *Trace* menu contains the following functions and submenus:

- *Measure* selects the quantity to be measured and displayed.
- *Format* defines how the measured data is presented in the graphical display.
- *Scale* defines how the current trace is presented in the diagram selected in the *Format* submenu.
- Trace Select provides functions to handle traces and diagram areas, and assign traces to channels.
- *Trace Funct(ions)* store traces to the memory and perform mathematical operations on traces.
- *Marker* positions markers on a trace, configures their properties and selects the format of the numerical readout.
- *Search* uses markers to locate specific points on the trace.
- *Marker Funct(ions)* define the sweep range, scale the diagram and introduce an electrical length offset using the active marker.
- *Lines* defines limits for measured values and activates the limit check.

## Measure

The *Measure* submenu selects the quantity to be measured and displayed.




- *S11,* S12, S21, S22 select the four elements of the standard 2-port scattering matrix (S-parameters).
- *More S-Parameters* opens a dialog to select S-parameters for multiport measurements or mixed mode S-parameters.
- *Ratios* opens a submenu to select and define ratios of wave quantities.
- *Wave Quantities* opens a submenu to select the absolute (unratioed) power of the analyzed signals (wave quantities).
- *Impedance* opens a submenu to convert reflection S-parameters into matched-circuit impedances (converted Z-parameters).
- *Admittance* opens a submenu to convert reflection S-parameters into matched-circuit admittances (converted Y-parameters).
- *Z-Parameters* opens a dialog to select arbitrary open-circuit impedance parameters describing a linear circuit.
- *Y-Parameters* opens a dialog to select arbitrary short-circuit admittance parameters describing a linear circuit.
- *Stability Factor* selects one of the three factors K, $\mu_1$ or $\mu_2$ to assess the stability of linear circuits (e.g. amplifiers).
- *Imbalance* opens a dialog to select the imbalance for balanced ports.
- *All S-Params* displays all S-parameters that the analyzer can measure in the appropriate diagram areas.
- *DC Inputs* selects and configures the measurement of DC powers and the Power Added Efficiency (PAE).
- *Power Sensor* opens a configuration dialog for the measurement of wave quantities using an external power meter.

**Port assignments of the DUT and the analyzer**

S-parameters $S_{<out><in>}$ characterize the Device Under Test, so the indices <out> and < in> denote the output and input ports of the DUT. Analogously the waves $a_1$ and $a_2$ are referred to as incident waves, $b_1$ and $b_2$ to as outgoing waves.

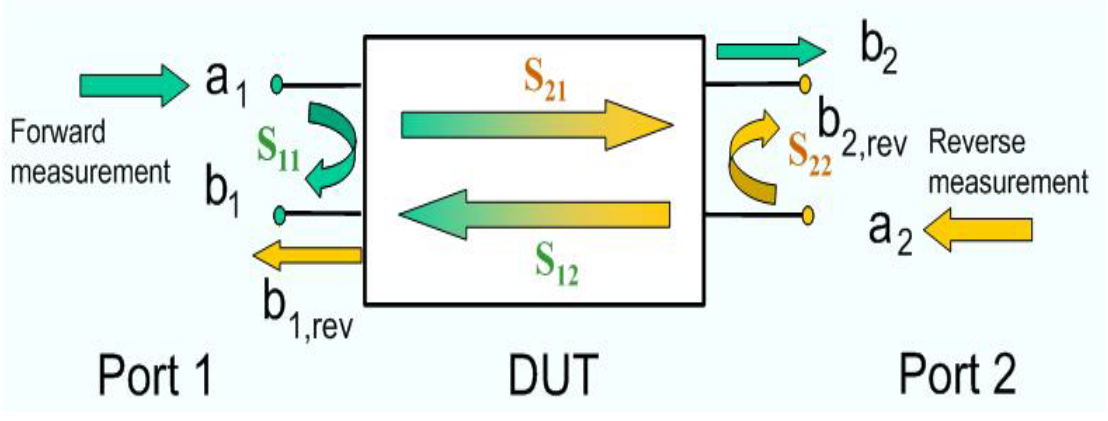
The analyzer measures the waves $a_1$, $b_1$, ... at its own test ports in order to obtain S-parameters, ratios and other derived quantities. The test ports of the analyzer are numbered, so it is convenient to use them as a reference, defining port number n of the DUT as the port connected to test port n of the analyzer. With this convention the waves $a_n$ and $b_n$ have the following meaning:

- $a_n$ is the wave transmitted at test port no. n of the analyzer (stimulus signal) and fed to port number n of the DUT (incident wave).
- $b_n$ is the wave transmitted at port number n of the DUT (response signal) and received at test port no. n of the analyzer (received wave).

*The analyzer offers various diagrams and display formats to visualize the results. The analyzer places no restriction on the formats that are available for each measured quantity. However, to keep the interpretation simple, it is recommended to select an appropriate display format; see Trace - Format.*

## S11, S12, S21, S22

Select one of the four elements of the standard 2-port scattering matrix (S-parameters).




The 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 are expressed as $S_{<out><in>}$, where <out> and <in> denote the output and input port numbers of the DUT.

A full 2-port S-parameter measurement involves 2 stages (referred to as partial measurements) with interchanged drive and receive ports. The analyzer automatically switches the internal power sources and the receivers to obtain the desired S-parameters.

**Remote control:**

```
CALCulate<Ch>:PARameter:MEASure "<Trace_Name>", "S11" | "S12" |
"S21" | "S22"
[SENSe<Chn>:]FUNCtion[:ON] "...:POWer:S<11 | 12 | 21 | 22>"
```

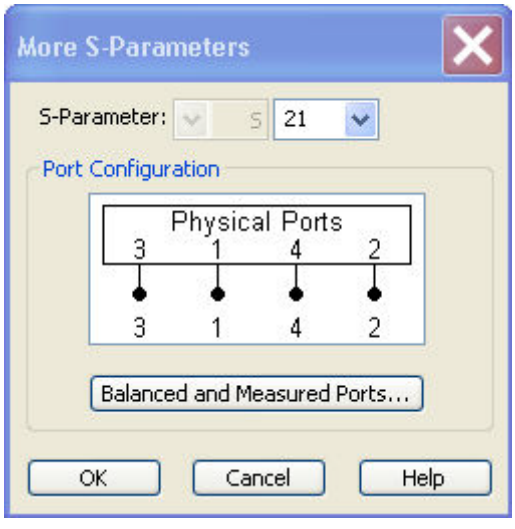
Create new trace and select name and measurement parameter:

```
CALCulate<Ch>:PARameter:SDEFine "<Trace_Name>", "S11" | "S12" |
"S21" | "S22"
```

Use `CALCulate<Ch>:PARameter:DEFine:SGRoup` to create traces for all S-parameters associated with a group of logical ports.

## More S-Parameters

Opens a dialog to select S-parameters for multiport measurements (including the 2-port S-parameters) or mixed mode S-parameters.




The *More S-Parameters* dialog provides the following settings:

- *S-Parameter* selects the type (left pull-down list) and the port number assignment (right pull-down list) of the *S-parameter*. Mixed mode parameters are only available if a balanced port configuration is active. They are expressed as $S_{<mout><min>}$, where <mout> and <min> denote the output and input port modes. The port numbers are assigned in the order $S_{<out><in>}$, where <out> and <in> denote the output (response) and input (stimulus) port numbers of the DUT.
- *Balanced and Measured Ports* opens the dialog of the same name to define the properties of the test ports. Single-ended (unbalanced) S-parameters are assigned to the physical test ports of the analyzer. Balanced S-parameters are assigned to logical test ports. Selecting a balanced port configuration with logical test ports means that the unbalance-balance conversion is switched on and that the analyzer provides mixed mode S-parameters.
- The graphics above the *Balanced and Measured Ports* button illustrates the current port configuration.

**The port configuration is valid for all traces in the channel**

The settings made in the *Balanced and Measured Ports* dialog are channel settings and therefore apply to all traces assigned to the channel *(Balanced and Measured Ports* is also accessible through *Channel – Mode– Port Config...*). Within a channel the analyzer measures either single-ended or mixed mode parameters. If a balanced test port configuration is selected the single-ended parameters assigned to the channel are converted into mixed mode parameters.

**Remote control:**

```
CALCulate<Ch>:PARameter:MEASure "<Trace_Name>", "S11" | "S12" |
"S21" | "S22"
[SENSe<Chn>:]FUNCtion[:ON] "...:POWer:S<11 | 12 | 21 | 22>"
```

Create new trace and select name and measurement parameter:

```
CALCulate<Ch>:PARameter:SDEFine "<Trace_Name>", "S11" | "S12" |
"S21" | "S22"
```

Use `CALCulate<Ch>:PARameter:DEFine:SGRoup` to create traces for all S-parameters associated with a group of logical ports.

## Single-Ended and Mixed Mode Parameters

The analyzer measures single-ended (unbalanced) parameters (S-parameters, impedances, admittances, Z-parameters, Y-parameters) unless a balanced port configuration is selected. In the *More S-Parameters* dialog and the analogous dialogs for the other quantities, the left parameter selection field shows the parameter type *(S, Y, Z)* and is disabled (grayed). The range of output and input port numbers <out> and <in> depends on the analyzer model.

The analyzer switches to mixed mode parameter measurement if a balanced port configuration is selected. The left parameter selection field is enabled and offers a selection of matrices $S_{<mout><min>}$ (or $Z_{<mout><min>}$, ...) where <mout> and <min> denote the output and input port modes. The following three output or input modes are available:

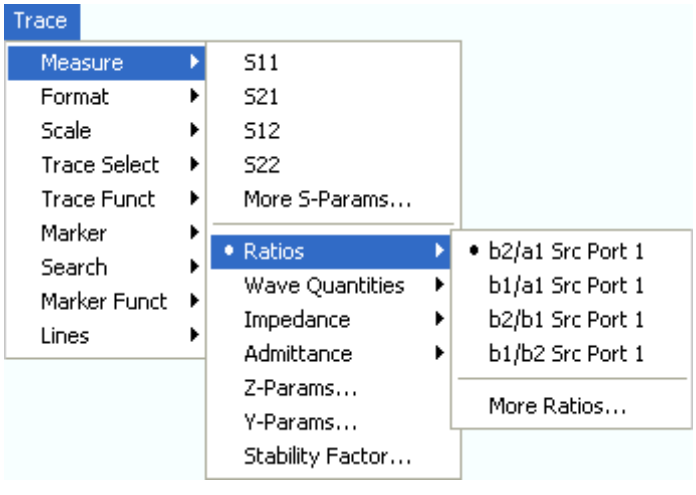
- s: Single-ended (for unbalanced ports)
- d: Differential mode (for balanced ports)
- c: Common mode (for balanced ports)

All possible combinations of mixed mode parameters (e.g. Sss, Scs, Sds, Sdd,..., Zss, Zcs,...) are provided. Sss is used for unbalanced S-parameters if the balanced-unbalanced conversion is active; otherwise the port mode indices ss are omitted.

The selected modes must be compatible with the port configuration. If an attempt is made to select an incompatible parameter (e.g. a single-ended parameter for a balanced port), the analyzer displays an error message.

## Ratios

The *Ratios* submenu selects and defines ratios of wave quantities.



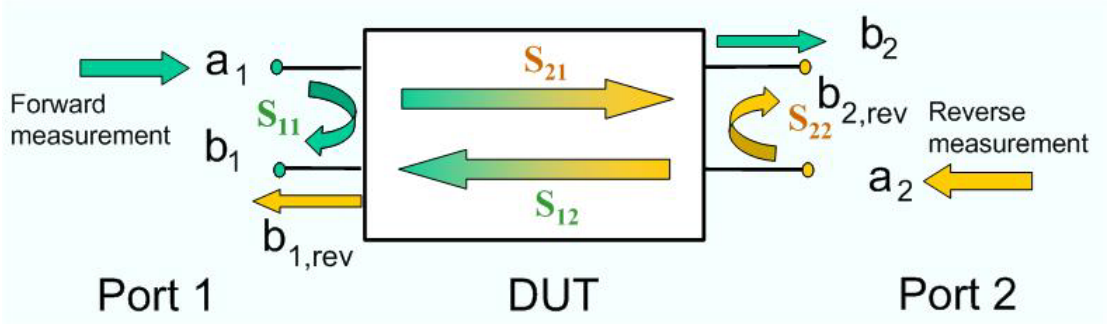

- *b2/a1 Src Port 1, b1/a1 Src Port 1, b2/b1 Src Port 1*, and *b1/b2 Scr Port 1* select predefined ratios of the standard 2-port wave quantities $a_1$, $a_2$, $b_1$, and $b_2$, to be measured in forward direction with port 1 providing the stimulus signal.

- *More Ratios* opens a dialog to define and select arbitrary ratios of wave quantities for different detectors and source ports or higher port numbers. In true differential mode, the dialog also offers ratios of balanced wave quantities.

In contrast to S-parameters, ratios are not system error corrected. A power calibration can be applied to ratios; see *Data Flow* in the *System Overview* chapter.

## b2/a1 Src Port 1, b1/a1 Src Port 1, b2/b1 Src Port 1, b1/b2 Src Port 1

Select predefined complex ratios of the standard 2-port wave quantities $a_1$, $a_2$, $b_1$, and $b_2$:




The predefined wave quantities are all obtained with the same test set configuration, port 1 providing the stimulus signal (source port 1, forward measurement if the stimulus signal is fed to the input of the DUT).

- *b2/a1 Src 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).
- *b1/a1 Src Port 1* is the ratio of the wave quantities $b_1$ and $a_1$, measured at PORT 1, and corresponds to the S-parameter $S_{11}$ (input reflection coefficient).
- *b2/b1 Src Port 1* is the ratio of the transmitted to the reflected wave quantities in a forward measurement.
- *b1/b2 Src Port 1* is the inverse of *b2/a1 Src Port 1*.

The analyzer can also measure arbitrary ratios for other source ports; see *More Ratios*.

**Remote control:**

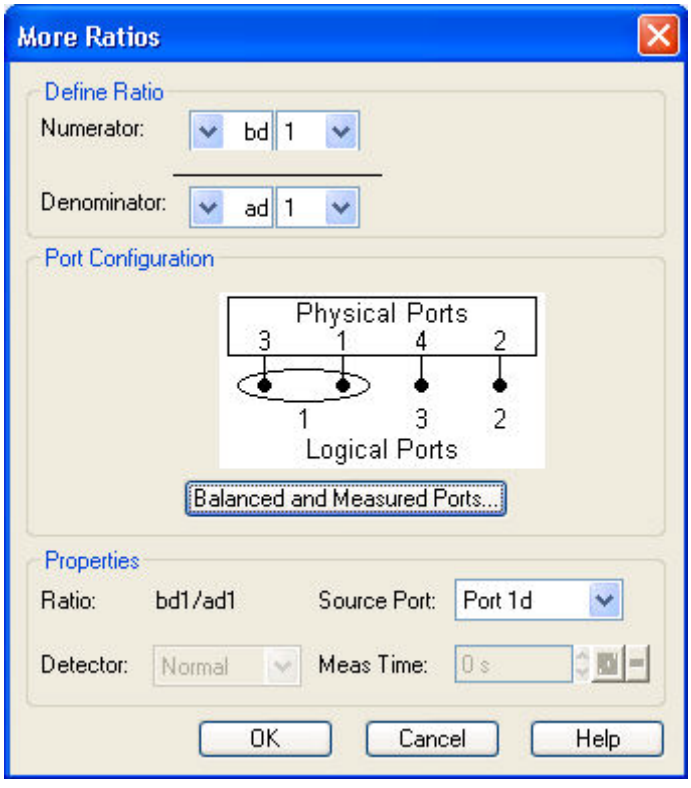
```
CALCulate<Ch>:PARameter:MEASure "<Trace_Name>", "B2/A1" | ...
[SENSe<Chn>:]FUNCtion[:ON] "...POWer:RATIO B2, A1" etc.
```

Create new trace and select name and measurement parameter:

```
CALCulate<Ch>:PARameter:SDEFine "<Trace_Name>", "B2/A1" | ...
```

## More Ratios

Opens a dialog to select arbitrary ratios of wave quantities, e.g. for different detectors and source ports or higher port numbers. In true differential mode, the dialog also offers ratios of balanced wave quantities.




The notation for ratios and the functionality of the *More Ratios* dialog is analogous to the definition of S-parameters.

- *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. Ratios of balanced wave quantities are only available if the True Differential Mode is active; see background information below. The input (stimulus) or output (response) port number is selected in the right pull-down list. The range of output and input port numbers depends on the analyzer model.
- *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. The same waves as for the *Numerator* are available.
- *Ratio* indicates the selected ratio.
- *Source Port* contains all analyzer ports or external generators which can be used as a source for the stimulus signal. The list contains all analyzer ports *Port 1* to *Port n*. Generators *(Gen 1, Gen 2, ...)* must be configured explicitly in the *System Configuration – External Generators* dialog before they appear in the list.
- *Detector* selects the algorithm that is used to calculate the displayed trace from the individual measurement points. In the *Max.* and *RMS* detector modes, it is possible to define the detector observation time *(Meas. Time)* for a single measurement point. The *Detector* and *Meas. Time* controls are both unavailable in true differential mode, because only the *Normal* detector provides consistent phase information. Detector settings are available for ZVA and ZVT network analyzers only.

**External generators**

The measurement process for external generators *Gen 1, Gen2...* in the *Source Port* list differs fromthe measurement process for internal source ports:

1. An external generator always represents a permanent signal source that is switched on for all partial measurements. In contrast, an analyzer port is switched off for partial measurements

that do not require a source signal. Exception: Internal ports can be configured as permanent sources (see *Gen* setting in the*Source* section of the *Port Configuration* dialog).

2. 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.

**Ratios of single-ended and balanced wave quantities**

If a balanced port configuration is selected and the True Differential Mode is active, the left *Numerator* and *Denominator* pull-down lists contain the following wave types:

- as, bs: Single-ended incident (a) or outgoing (b) wave. To select this wave type, the selected port must be unbalanced. If the port configuration contains no unbalanced ports, the single-ended wave quantities are hidden.
- ad, ac, bd, bc: Balanced incident (a) or outgoing (b) waves. d and c distinguish between the differential and common mode; see Wave Quantities and Ratios.

**Remote control:**

```
CALCulate<Ch>:PARameter:MEASure "<Trace_Name>", "B2/A1" | ...
[SENSe<Chn>:]FUNCtion[:ON] "...POWer:RATIO B2, A1" etc.
```

Create new trace and select name and measurement parameter:

```
CALCulate<Ch>:PARameter:SDEFine "<Trace_Name>", "B2/A1" | ...
```

## Detector (for R&S ZVA and R&S ZVT Analyzers)

Selects the algorithm that is used to calculate the displayed measurement points from the raw data. The *Detector* can be selected in the *More Ratios* and *More Wave Quantities* dialogs. Detector settings are available for ZVA and ZVT network analyzers only.

The following detectors are available:

- Normal selects the default detector mode where each measurement point is displayed without modification as soon as it is recognized to be valid. 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.
- Peak collects all valid results at each sweep point during the *Meas. Time* set in the *More Ratios* or *More Wave Quantities* dialog and selects the result with the maximum magnitude (maximum power) as the measurement point. 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.
- RMS collects all valid results at each sweep point during the *Meas. Time* set in the *More Ratios* or *More Wave Quantities* dialog and calculates the root mean square (RMS) of the linear magnitude of these results. This yields the magnitude of the measurement point. 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 detector settings have an influence on the displayed trace. They do not affect the number of measurement points in a sweep (or in a subset of the sweep range) but may increase the sweep time In the *Peak* and *RMS* detector settings, the sweep time can not be smaller than the detector observation time *(Meas. Time)* set in the *More Ratios* or *More Wave Quantities* dialogs. The sweep time can be larger if additional delays are taken into account. Changing the *Meas. Time* affects the sweep time. On the other hand, increasing the *Sweep Time* in the *Channel – Sweep* menu leaves the *Meas. Time* unchanged .

**Combining different detectors**

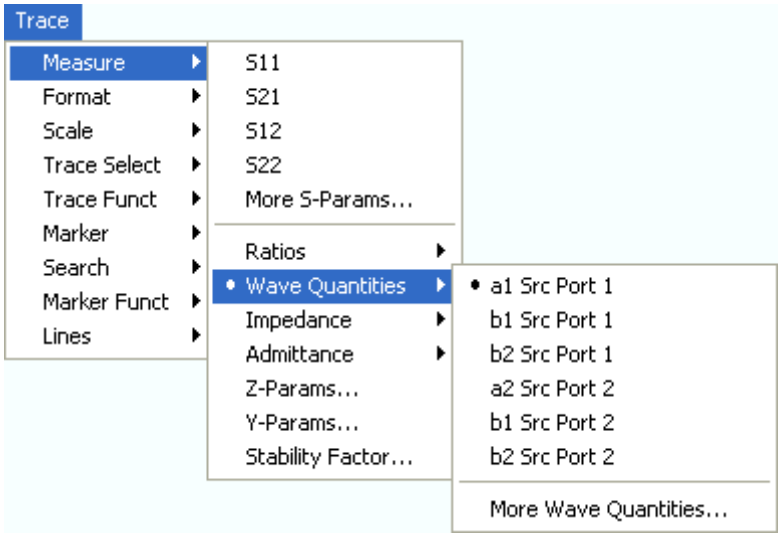
The detector setting in the *More Ratios* menu affects the ratio of a numerator and a denominator wave quantity. This does not place any restriction on the measurement functionality of the analyzer, because ratios can be formed by calculating the numerator and denominator individually and using the trace functions. A possible application is the comparison of different detector settings for a particular trace.

*The analyzer generates a warning if the selected measurement time for Peak or RMS 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 RMS detectors is too short. Increase the measurement time and/or increase the IF bandwidth until the warning disappears.*

**Remote control:**
```
CALCulate<Ch>:PARameter:MEASure "<Trace_Name>", "<Par>"
CALCulate<Ch>:PARameter:SDEFine "<Trace_Name>", "<Par>"
```
(the parameter name <Par> also contains the detector)
```
[SENSe<Ch>:]SWEep:DETector:TIME
```

## Wave Quantities

The *Wave Quantities* submenu selects a wave quantity to be measured and displayed.

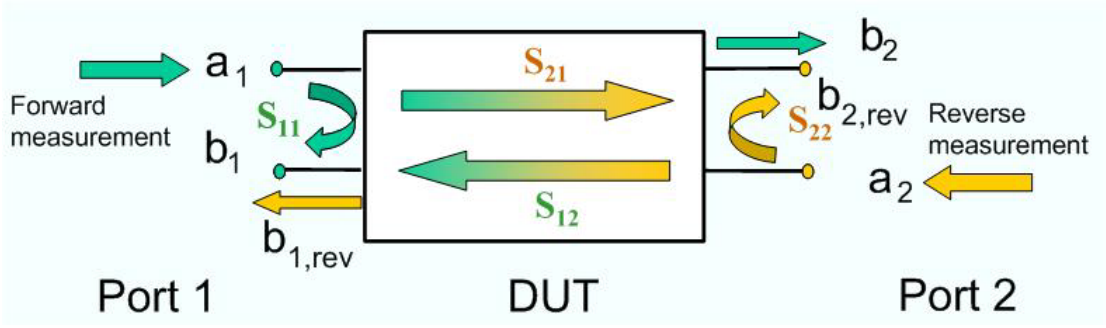



- *a1 Src Port 1, b1 Src Port 1, b2 Src Port 1, a2 Src Port 2, b1 Src Port 2* and *b2 Src Port 2* select the standard 2-port wave quantities $a_1$, $b_1$, $a_2$, and $b_2$, to be measured in forward or reverse direction with port 1 or 2 providing the stimulus signal.
- *More Wave Quantities* opens a dialog to define and select arbitrary wave quantities for different detectors and source ports or higher port numbers. In true differential mode, the dialog also offers balanced wave quantities.

In contrast to S-parameters, wave quantities are not system error corrected. A power calibration can be applied to wave quantities; see *Data Flow* in the *System Overview* chapter.

### a1 Src Port 1, b1 Src Port 1, b2 Src Port 1, a2 Src Port 2, b1 Src Port 2, b2 Src Port 2

Select the standard 2-port wave quantities $a_1$, $a_2$, $b_1$, and $b_2$ for different source ports:




The predefined wave quantities are obtained with different source ports. *a1 Src Port 1, b1 Src Port 1* and *b1 Src Port 2* are measured at Port 1 of the analyzer. *a2 Src Port 2, b2 Src Port 1* and *b2 Src Port 2* are measured at Port 2 of the analyzer.

- *a1 Src Port 1* is the wave transmitted at test port 1. In a standard S-parameter measurement, this wave is fed to the input port (port 1) of the DUT (forward measurement).
- *b1 Src Port 1* is the wave received at test port 1. In a standard S-parameter measurement, this is the reflected wave at port 1 of the DUT (forward measurement).
- *b2 Src Port 1* is the wave received at test port 2. In a standard S-parameter measurement, this wave is transmitted at port 2 of the DUT (forward measurement).
- *a2 Src Port 2* is the wave transmitted at test port 2. In a standard S-parameter measurement, this wave is fed to the output port (port 2) of the DUT (reverse measurement).
- *b1 Src Port 2* is the wave received at test port 1. In a standard S-parameter measurement, this wave is transmitted at port 2 of the DUT (reverse measurement).
- *b2 Src Port 2* is the wave received at test port 2. In a standard S-parameter measurement, this wave is fed to the output port (port 2) of the DUT (reverse measurement).

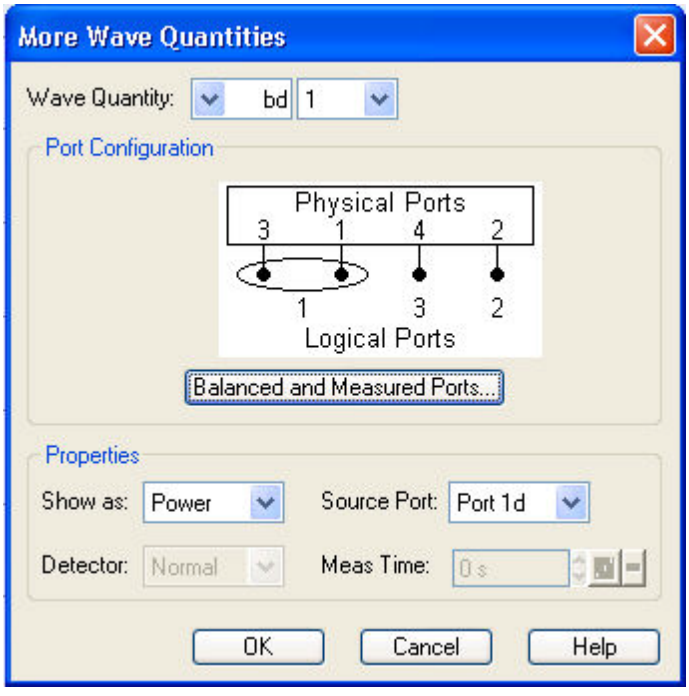
The analyzer can also measure wave quantities for other source ports; see *More Wave Quantities*.

**Remote control:** `CALCulate<Ch>:PARameter:MEASure "<Trace_Name>", "A1" | ...`
`[SENSe<Chn>:]FUNCtion[:ON] "...POWer:A1"` etc.

Create new trace and select name and measurement parameter:
`CALCulate<Ch>:PARameter:SDEFine "<Trace_Name>", "A1" | ...`

## More Wave Quantities

Opens a dialog to define and select arbitrary wave quantities for different detectors and source ports or higher port numbers. In true differential mode, the dialog also offers balanced wave quantities.




The notation for wave quantities and the functionality of the *More Wave Quantities* dialog is analogous to the definition of S-parameters.

- *Wave Quantity* selects the type (left pull-down list) and the port number assignment (right pull-down list) of the wave quantity. Balanced wave quantities are only available if the True Differential Mode is active; see background information below. The output (transmit) or input (receive) port number of the analyzer (corresponding to the input (stimulus) or output (response) port number of the DUT, respectively) is selected in the right pull-down list. The range of output and input port numbers depends on the analyzer model.
- *Show as* selects the physical unit of the displayed trace. It is possible to display the measured *Voltage* U or convert the wave quantity into an effective power according to $P = V^2/\mathrm{Re}(Z_0)$. $Z_0$ 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 *Port Configuration* dialog. This function is also available for memory traces where the remaining control elements of the dialog are grayed.
- *Source Port* contains all analyzer ports or external generators which can be used as a source for the stimulus signal. The list contains all analyzer ports *Port 1* to *Port n*. Generators *(Gen 1, Gen 2, ...)* must be configured explicitly in the *System Configuration – External Generators* dialog before they appear in the list. The analyzer places no restriction on the combination of source ports and port numbers of the measured wave quantity, so it is even possible to measure $a_2$ while the source port is port 1 (e.g. in order to estimate the directivity of the coupler in the internal test set).
- *Detector* selects the algorithm that is used to calculate the displayed trace from the individual measurement points. In the *Max.* and *RMS* detector modes, it is possible to define the observation time *(Meas. Time)* for a single measurement point. The *Detector* and *Meas. Time* controls are both unavailable in true differential mode, because only the *Normal* detector provides consistent phase information. Detector settings are available for ZVA and ZVT network analyzers only.

**External generators**

The measurement process for external generators *Gen 1, Gen2...* in the *Source Port* list differs fromthe measurement process for internal source ports:

3. An external generator always represents a permanent signal source that is switched on for all partial measurements. In contrast, an analyzer port is switched off for partial measurements that do not require a source signal. Exception: Internal ports can be configured as permanent sources (see *Gen* setting in the*Source* section of the *Port Configuration* dialog).
4. 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.

**Single-ended and balanced wave quantities**

If a balanced port configuration is selected and the True Differential Mode is active, the left *Wave Quantity* pull-down list contains the following wave types:

- as, bs: Single-ended incident (a) or outgoing (b) wave. To select this wave type, the selected port must be unbalanced. If the port configuration contains no unbalanced ports, the single-ended wave quantities are hidden.
- ad, ac, bd, bc: Balanced incident (a) or outgoing (b) waves. d and c distinguish between the differential and common mode; see Wave Quantities and Ratios.

**Remote control:** `CALCulate<Ch>:PARameter:MEASure "<Trace_Name>", "A1" | ...`
`[SENSe<Chn>:]FUNCtion[:ON] "...POWer:A1"` etc.

Create new trace and select name and measurement parameter:
`CALCulate<Ch>:PARameter:SDEFine "<Trace_Name>", "A1" | ...`

## Impedance

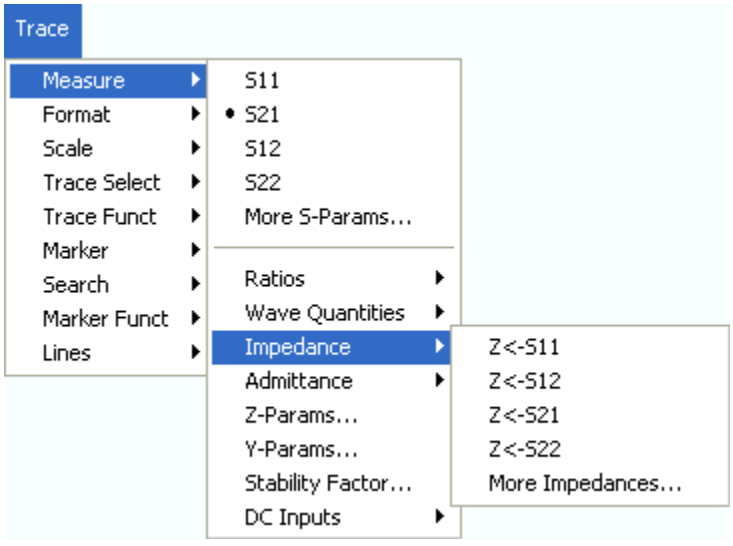
The *Impedance* submenu contains the functions to convert S-parameters into matched-circuit impedances.

**Matched-circuit impedances and open-circuit Z-parameters**

The analyzer provides two different types of impedance parameters that are essentially based on different n-port circuit models:

- The matched-circuit impedances describe the impedances of a DUT that is terminated at its outputs with the reference impedance $Z_0$.
- The open-circuit Z-parameters describe the impedances of a DUT with open output ports (I = 0). Open-circuit parameters can be selected in the *More Z-Parameters* dialog.

Mixed mode measurements are available for both types of impedance parameters.




- *Z <– S11, Z <– S12, Z <– S21, Z<– S22* select the forward and reverse matched-circuit impedances of a 2-port DUT.
- *More Impedances...* opens a dialog to define and select matched-circuit impedances for more ports or mixed mode measurements.

## Z <– S11, Z <– S12, Z <– S21, Z <– S22

Select the 2-port matched-circuit, converted impedances. 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 of a 2-port DUT that is terminated at its output with the reference impedance $Z_0$ (matched-circuit impedance measured in a forward reflection measurement).
- $Z_{22}$ is the output impedance of a 2-port DUT that is terminated at its input 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 transfer impedances, respectively.

The analyzer can also provide matched-circuit impedances for more drive ports or balanced port configurations; see *More Impedances*.

Use the *More Z-Parameters* dialog to measure open-circuit Z-parameters. Use the Smith chart to obtain an alternative, graphical representation of the converted impedances in a reflection measurement.

**Remote control:**

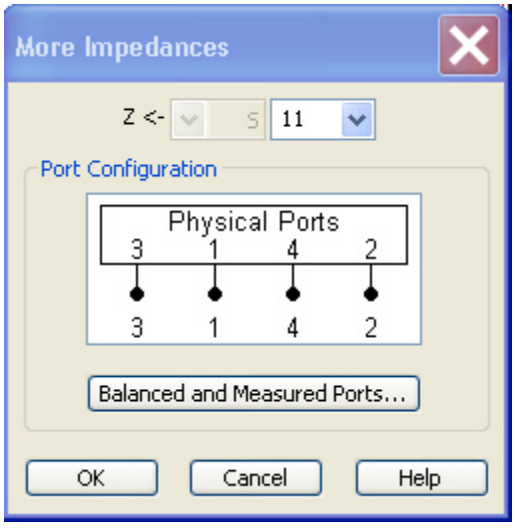
```
CALCulate<Ch>:PARameter:MEASure "<Trace_Name>", "Z-S11" | "Z-S12" | "Z-S21" | "Z-S22"
[SENSe<Chn>:]FUNCtion[:ON] "...:POWer:Z<11 | 12 | 21 | 22>"
```

Create new trace and select name and measurement parameter:

```
CALCulate<Ch>:PARameter:SDEFine "<Trace_Name>", "Z-S11" | "Z-S12" | "Z-S21" | "Z-S22"
```

## More Impedances

Opens a dialog to select converted, matched-circuit impedance parameters for more ports or balanced port measurements.




The notation for converted impedance parameters and the functionality of the *More Impedances* dialog is analogous to the definition of S-parameters.

- *Z*<— selects the type (left pull-down list) and the port number assignment (right pull-down list) of the impedance parameter. Mixed mode parameters are only available if a balanced port configuration is active. They are expressed as $Y_{<mout><min>}$, where <mout> and <min> denote the output and input port modes. The port numbers are assigned in the order $Y_{<out>< in>}$, where <out> and <in> denote the output (response) and input (stimulus) port numbers of the DUT.
- *Balanced and Measured Ports* opens the dialog of the same name to define the properties of the test ports. Single-ended (unbalanced) impedance parameters are assigned to the physical test ports of the analyzer. Balanced impedance parameters are assigned to logical test ports. Selecting a balanced port configuration with logical test ports means that the unbalance-balance conversion is switched on and that the analyzer provides mixed mode parameters.
- The graphics above the *Balanced and Measured Ports* button illustrates the current port configuration.

**The port configuration is valid for all traces in the channel**

The settings made in the *Balanced and Measured Ports* dialog are channel settings and therefore apply to all traces assigned to the channel *(Balanced and Measured Ports* is also accessible through *Channel – Mode– Port Config...*). Within a channel the analyzer measures either single-ended or mixed mode parameters. If a balanced test port configuration is selected the single-ended parameters assigned to the channel are converted into mixed mode parameters.

**Relation between impedances and S-parameters**

The relation between the generalized (multiport and mixed mode) impedance parameters and the S-parameters is analogous to the 2-port case. For reflection measurements:

$$Z_{<out><in>ii} = Z_0 \frac{1+S_{<out><in>ii}}{1-S_{<out><in>ii}}, \quad <out> = <in>,$$

and for transmission measurements:

$$Z_{<out><in>ij} = 2 \cdot \frac{\sqrt{Z_{0i} \cdot Z_{0j}}}{S_{<out><in>ij}} - \left(Z_{0i} + Z_{0j}\right), \quad i \neq j,$$

where $Z_{0i}$ denotes the reference impedance of the analyzer port no. i.

**Remote control:**

```
CALCulate<Ch>:PARameter:MEASure "<Trace_Name>", "Z-S11" | "Z-S12" ...
```

Create new trace and select name and measurement parameter:

```
CALCulate<Ch>:PARameter:SDEFine "<Trace_Name>", "Z-S11" | "Z-S22" | ...
```

## Admittance

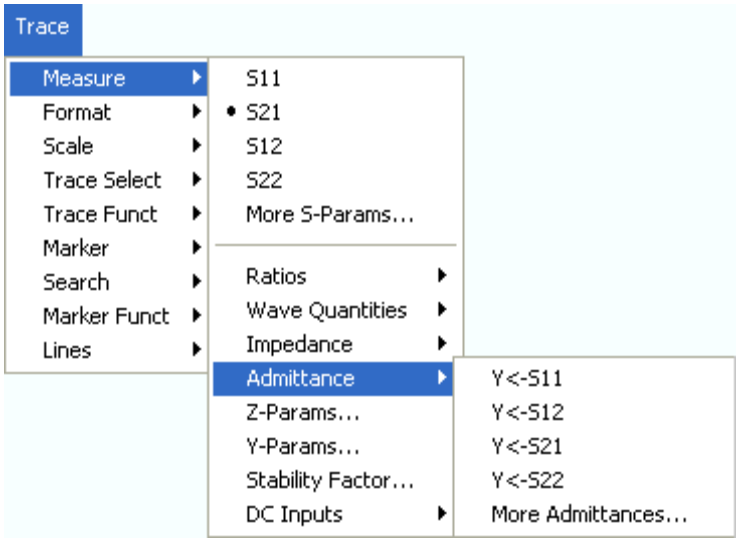
The *Admittance* submenu contains the functions to convert reflection S-parameters into matched-circuit admittances.

**Matched-circuit admittances and short-circuit Y-parameters**

The analyzer provides two different types of admittance parameters that are essentially based on different n-port circuit models:

- The matched-circuit admittances describe the admittances of a DUT that is terminated at its outputs with the reference impedance $Z_0$.
- The short-circuit Y-parameters describe the admittances of a DUT with output ports terminated in a short circuit (V = 0). Short-circuit Y-parameters can be selected in the *More Y-Parameters* dialog.

Mixed mode measurements are available for both types of admittance parameters.




- *Y <– S11, Y <– S12, Y <– S21, Y <– S22* select the forward or reverse matched-circuit admittances of a 2-port DUT.
- *More Admittances*... opens a dialog to define and select short-circuit admittances for more ports or mixed mode measurements.

### Y <– S11, Y <– S12, Y <– S21, Y <– S22

Select the 2-port converted matched-circuit 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 of a 2-port DUT that is terminated at its output with the reference impedance $Z_0$ (matched -circuit admittance measured in a forward reflection measurement).
- $Y_{22}$ is the output admittance of a 2-port DUT that is terminated at its input 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 transfer admittances, respectively.

The analyzer can also provide matched-circuit admittances for more drive ports or balanced port configurations; see More Admittances.

Use the *More Y-Parameters* dialog to measure short-circuit Y-parameters including the transfer parameters. Use the Inverted Smith chart to obtain an alternative, graphical representation of the converted admittances in a reflection measurement.

**Remote control:**

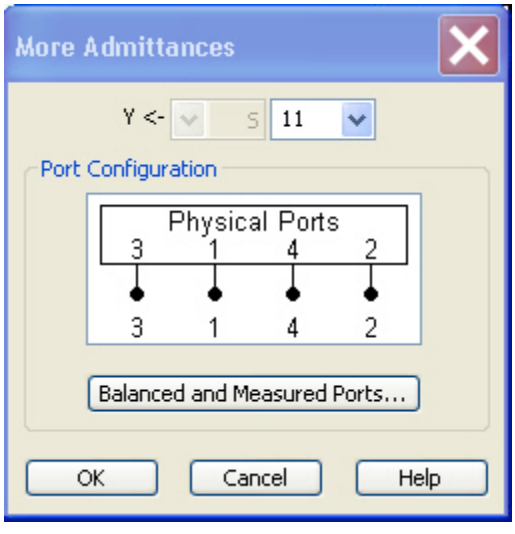
```
CALCulate<Ch>:PARameter:MEASure "<Trace_Name>", "Y-S11" | "Y-S12" | "Y-S21" | "Y-S22"
[SENSe<Chn>:]FUNCtion[:ON] "...:POWer:Y<11 | 12 | 21 | 22>"
```

Create new trace and select name and measurement parameter:

```
CALCulate<Ch>:PARameter:SDEFine "<Trace_Name>", "Y-S11" | "Y-S12" | "Y-S21" | "Y-S22"
```

## More Admittances

Opens a dialog to select converted, matched-circuit admittances for more ports or balanced port measurements.




The notation for admittance parameters and the functionality of the *More Admittances* dialog is analogous to the definition of S-parameters.

- *Y<—* selects the type (left pull-down list) and the port number assignment (right pull-down list) of the admittance parameter. Mixed mode parameters are only available if a balanced port configuration is active. They are expressed as $Y_{<mout><min>}$, where <mout> and <min> denote the output and input port modes. The port numbers are assigned in the order $Y_{<out>< in>}$, where <out> and <in> denote the output (response) and input (stimulus) port numbers of the DUT.
- *Balanced and Measured Ports* opens the dialog of the same name to define the properties of the test ports. Single-ended (unbalanced) admittance parameters are assigned to the physical

test ports of the analyzer. Balanced admittance parameters are assigned to logical test ports. Selecting a balanced port configuration with logical test ports means that the unbalance-balance conversion is switched on and that the analyzer provides mixed mode parameters.

- The graphics above the *Balanced and Measured Ports* button illustrates the current port configuration.

**The port configuration is valid for all traces in the channel**

The settings made in the *Balanced and Measured Ports* dialog are channel settings and therefore apply to all traces assigned to the channel *(Balanced and Measured Ports* is also accessible through *Channel – Mode– Port Config...*). Within a channel the analyzer measures either single-ended or mixed mode parameters. If a balanced test port configuration is selected the single-ended parameters assigned to the channel are converted into mixed mode parameters.

The relation between the generalized (multiport and mixed mode) admittance parameters and the S-parameters is analogous to the 2-port case and follows from the relations for impedance parameters.

**Remote control:**

```
CALCulate<Ch>:PARameter:MEASure "<Trace_Name>", "Y-S11" | "Y-S12" ...
```

Create new trace and select name and measurement parameter:

```
CALCulate<Ch>:PARameter:SDEFine "<Trace_Name>", "Y-S11" | "Y-S22" | ...
```

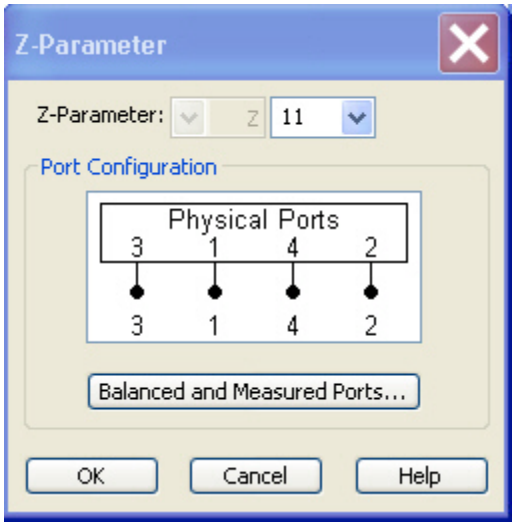
## Z-Parameters...

Opens the *Z-Parameter* dialog to select arbitrary open-circuit Z-parameters for different ports or mixed mode measurements. In analogy to the matched-circuit impedances, the open-circuit Z-parameters are available for any combination of input and output ports.

Open-circuit Z-parameters can be used as an alternative to S-parameters (or short-circuit Y-parameters) in order to completely characterize a linear n-port network.

### Z-Parameter

Is a dialog to select open-circuit Z-parameters for multiport measurements (including the 2-port Z-parameters) or mixed mode Z-parameters.




The notation for Z-parameters and the functionality of the *Z-Parameter* dialog is analogous to the definition of S-parameters. The dialog provides the following settings:

- *Z-Parameter* selects the type (left pull-down list) and the port number assignment (right pull-down list) of the Z-parameter. Mixed mode parameters are only available if a balanced port configuration is active. They are expressed as $Z_{<mout><min>}$, where <mout> and <min> denote the output and input port modes. The port numbers are assigned in the order $Z_{<out>< in>}$, where <out> and <in> denote the output (response) and input (stimulus) port numbers of the DUT.
- *Balanced and Measured Ports* opens the dialog of the same name to define the properties of the test ports. Single-ended (unbalanced) Z-parameters are assigned to the physical test ports of the analyzer. Balanced Z-parameters are assigned to logical test ports. Selecting a balanced port configuration with logical test ports means that the unbalance-balance conversion is switched on and that the analyzer provides mixed mode Z-parameters.
- The graphics above the *Balanced and Measured Ports* button illustrates the current port configuration.

**The port configuration is valid for all traces in the channel**

The settings made in the *Balanced and Measured Ports* dialog are channel settings and therefore apply to all traces assigned to the channel *(Balanced and Measured Ports* is also accessible through *Channel – Mode– Port Config...*). Within a channel the analyzer measures either single-ended or mixed mode parameters. If a balanced test port configuration is selected the single-ended parameters assigned to the channel are converted into mixed mode parameters.

**Remote control:**

```
CALCulate<Ch>:PARameter:MEASure "<Trace_Name>", "Z11" | "Z12" | "Z21" | "Z22"
```

Create new trace and select name and measurement parameter:

```
CALCulate<Ch>:PARameter:SDEFine "<Trace_Name>", "Z11" | "Z12" | "Z21" | "Z22"
```

## Y-Parameters

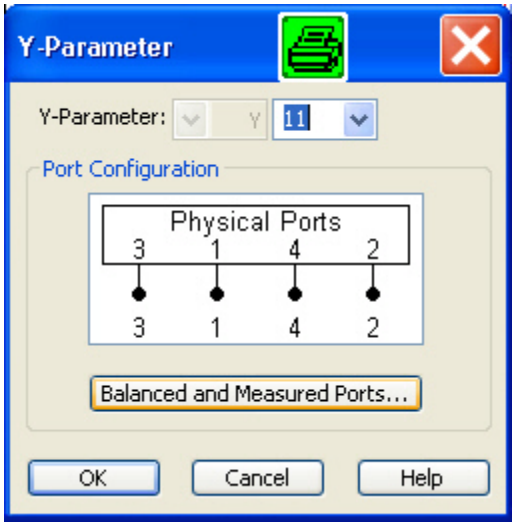
Opens the *Y-Parameter* dialog to select arbitrary short-circuit Y-parameters for different ports or mixed mode measurements. In analogy to the matched-circuit admittances, the short-circuit Y-parameters are available for any combination of input and output ports.

Short-circuit Y-parameters can be used as an alternative to S-parameters (or open-circuit Z-

parameters) in order to completely characterize a linear n-port network.

## Y-Parameter

Opens a dialog to select short-circuit Y-parameters for multiport measurements (including the 2-port Y-parameters) or mixed mode Y-parameters.




The notation for Y-parameters and the functionality of the *Y-Parameter* dialog is analogous to the definition of S-parameters. The dialog provides the following settings:

- *Y-Parameter* selects the type (left pull-down list) and the port number assignment (right pull-down list) of the Y-parameter. Mixed mode parameters are only available if a balanced port configuration is active. They are expressed as $Y_{<mout><min>}$, where <mout> and <min> denote the output and input port modes. The port numbers are assigned in the order $Y_{<out><in>}$, where <out> and <in> denote the output (response) and input (stimulus) port numbers of the DUT.
- *Balanced and Measured Ports* opens the dialog of the same name to define the properties of the test ports. Single-ended (unbalanced) Y-parameters are assigned to the physical test ports of the analyzer. Balanced Y-parameters are assigned to logical test ports. Selecting a balanced port configuration with logical test ports means that the unbalance-balance conversion is switched on and that the analyzer provides mixed mode parameters.
- The graphics above the *Balanced and Measured Ports* button illustrates the current port configuration.

**The port configuration is valid for all traces in the channel**

The settings made in the *Balanced and Measured Ports* dialog are channel settings and therefore apply to all traces assigned to the channel *(Balanced and Measured Ports* is also accessible through *Channel – Mode– Port Config...*). Within a channel the analyzer measures either single-ended or mixed mode parameters. If a balanced test port configuration is selected the single-ended parameters assigned to the channel are converted into mixed mode parameters.

**Remote control:**

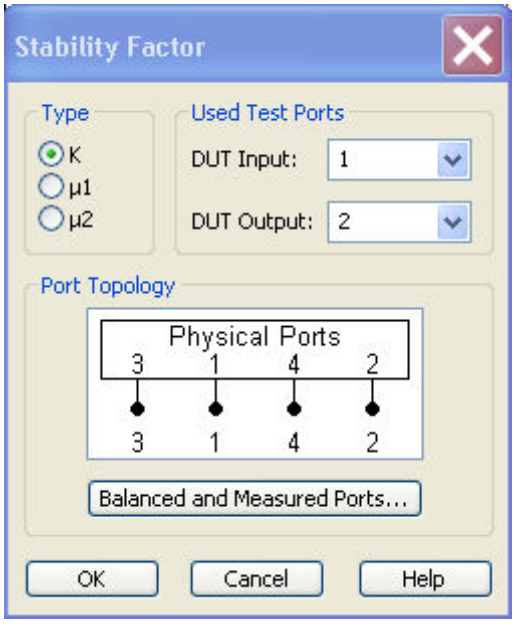
```
CALCulate<Ch>:PARameter:MEASure "<Trace_Name>", "Y11" | "Y12" | "Y21" | "Y22"
```

Create new trace and select name and measurement parameter:

```
CALCulate<Ch>:PARameter:SDEFine "<Trace_Name>", "Y11" | "Y12" | "Y21" | "Y22"
```

## Stability Factor...

Opens a dialog to select one of the three two port stability factors K, $\mu_1$ or $\mu_2$. Stability factors cannot be calculated in balanced port configurations.




The *Stability* dialog provides the following settings:

- *Stability Factor* selects the stability factor to be calculated; see definitions below.
- *Used Test Ports* selects the test port number of the analyzer to be connected to the input *(DUT Input)* and the output of the DUT *(DUT Output)*. The ports can be arbitrary, however, 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 *Port Topology* panel is analogous to the corresponding panel in the *More S Parameters* dialog. Stability factors can be calculated for unbalanced ports only. If a balanced port configuration is selected, the analyzer generates an error message.

Values of any of the stability factors less than 1 indicate potential instability.

**Definition of stability factors and stability criteria**

The stability factors K, $\mu_1$ and $\mu_2$ 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 by means of the stability factors $\mu_1$ and $\mu_2$.
- The $\mu_1$ and $\mu_2$ factors both provide a necessary and sufficient condition for unconditional stability: The conditions $\mu_1$>1 or $\mu_2$>1 are both equivalent to unconditional stability. This means that $\mu_1$ and $\mu_2$ 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.

**Remote control:**

```
CALCulate<Ch>:PARameter:MEASure "<Trace_Name>", "KFAC21" |
"MUF121" | "MUF221" | ...
[SENSe<Chn>:]FUNCtion[:ON] "...:POWer:KFACtor | MUFactor1 |
MUFactor2"
```

Create new trace and select name and measurement parameter:

```
CALCulate<Ch>:PARameter:SDEFine "<Trace_Name>", "SY11" | "SY12"
| "SY21" | "SY22"
```

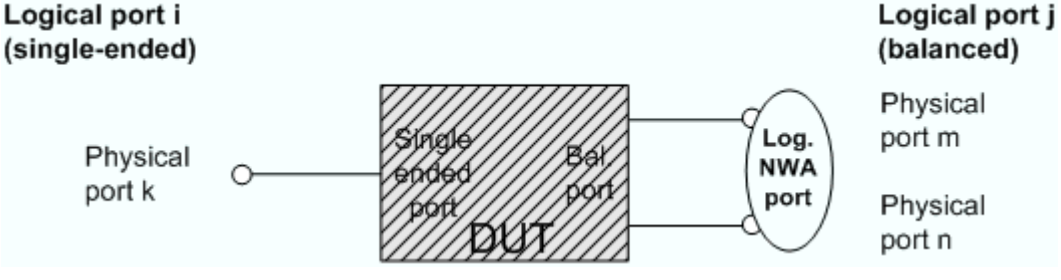
## Imbalance...

Opens a dialog to select the imbalance for balanced ports. This result is available in the simulated (virtual) differential mode but not in true differential mode.

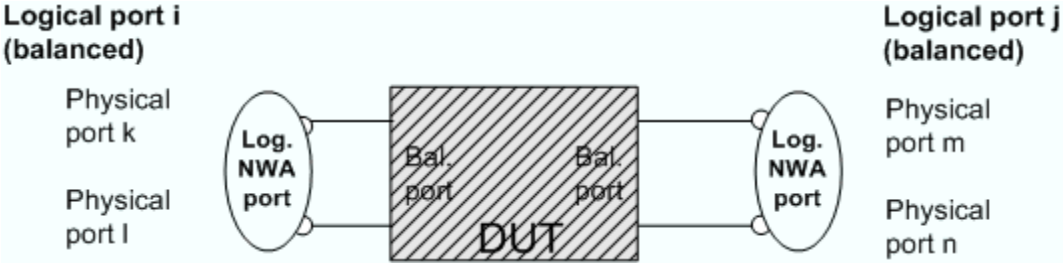
**Definition of imbalance parameters**

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 k, unbalanced signals with equal amplitude and opposite phase appear at the physical ports m and n.

This means that the ratio $-S_{km}/S_{kn}$ of the physical transmission coefficients of an ideal balun equals to 1. This ratio is called imbalance; it is a measure for the deviation of the balun from ideality. The definition of the imbalance of a DUT with one or two balanced ports and physical port numbers m < n, k < l is given below.

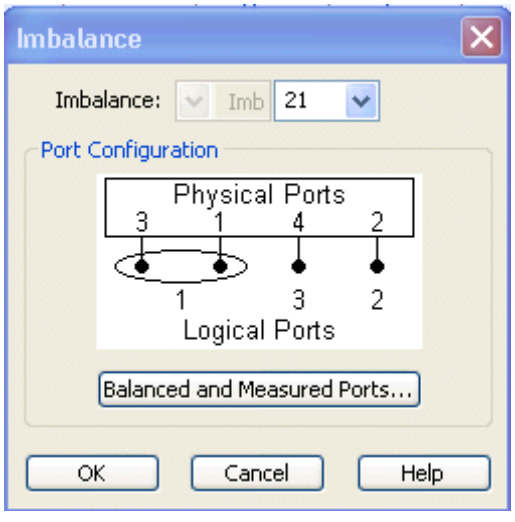



- The imbalance of a DUT with a single ended logical input port i and a balanced logical output port j is defined as Imb ij = $-S_{km}/S_{kn}$ and Imb ji = $-S_{mk}/S_{nk}$.




- The imbalance of a DUT with a balanced logical input port i and a balanced logical output port j is defined as Imb ij = $-(S_{km} - S_{kn})/(S_{lm} - S_{ln})$ and Imb ji = $-(S_{mk} - S_{nk})/(S_{ml} - S_{nl})$.

In general the imbalance is a quantity with two numeric indices numbering the logical output port and the logical input port of the DUT during the measurement (Imb<out><in>).




The *Imbalance* dialog provides the following settings:

- *Imbalance* selects the imbalance parameter *Imb*$_{<out><in>}$ to be measured.
- *Balanced and Measured Ports* opens the dialog of the same name to define the properties of the test ports. Single-ended (unbalanced) impedance parameters are assigned to the physical test ports of the analyzer. Balanced impedance parameters are assigned to logical test ports. Selecting a balanced port configuration with logical test ports means that the unbalance-balance conversion is switched on and that the analyzer provides mixed mode parameters. A unbalance-balance conversion is a prerequisite for the calculation of imbalance parameters.

**Remote control:**

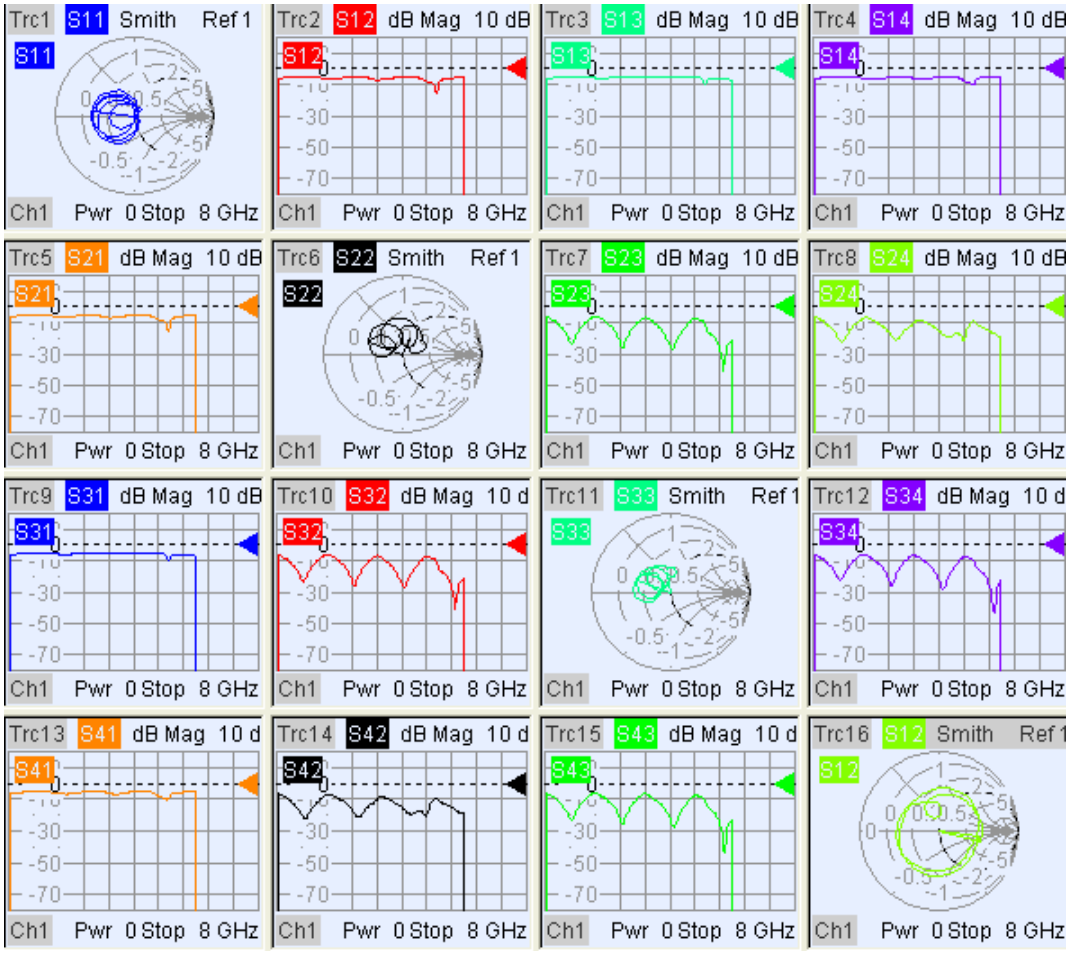
```
CALCulate<Ch>:PARameter:MEASure "<Trace_Name>", "IMB21" |
"IMB12" ...
```

Create new trace and select name and measurement parameter:

```
CALCulate<Ch>:PARameter:SDEFine "<Trace_Name>", "IMB21" |
"IMB12" ...
```

## All S-Params

Displays all S-parameters $S_{<out><in>}$ that the analyzer can calculate according to its port configuration. Reflection parameters $S_{ii}$ are displayed in Smith charts; transmission parameters are displayed in Cartesian (dB Mag) diagrams. For a four-port analyzer with four single-ended ports:

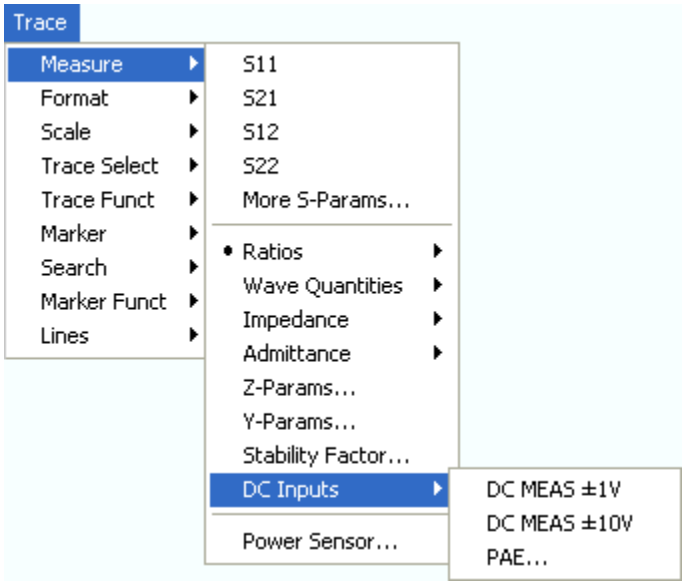



A n-port analyzer provides $n^2$ independent S-parameters $S_{<out><in>}$. If a balanced port configuration is defined, the normal S-parameters are replaced by mixed mode parameters.

**Remote control:** –

## DC Inputs

The *DC Inputs* submenu selects DC voltages or the Power Added Efficiency (PAE) of an active two-port DUT to be measured and displayed.




- DC MEAS ±1V and *DC MEAS ± 10V* select the DC voltages fed to the input connectors *DC MEAS* as measured quantities.
- PAE... opens a dialog to select and configure the PAE measurement.

All DC and PAE measurements use the input connectors *DC MEAS* at the rear of the instrument. The measurement results are real-valued and displayed as a function of sweep variable (frequency, internal source power, time). The measurements can be performed in parallel to all RF measurements.

## DC MEAS 1 V, DC MEAS 10 V

Select the DC voltages fed to the *DC MEAS* input connectors as measured quantities. The input connectors are located at the rear panel of the instrument.

The two DC inputs cover different input voltage ranges; see data sheet or rear panel labelling.

- The left input connector *DC MEAS ± 1V* provides the most accurate measurement for smaller voltages but has a restricted input level range.
- The right input connector *DC MEAS ± 10V* can be used for larger voltages.

**Remote control:**

```
CALCulate<Ch>:PARameter:MEASure "<Trace_Name>", "DC+1V" |
"DC+10V"
[SENSe<Chn>:]FUNCtion[:ON] "...:VOLTage[:DC] DCIN1 | DCIN2"
```

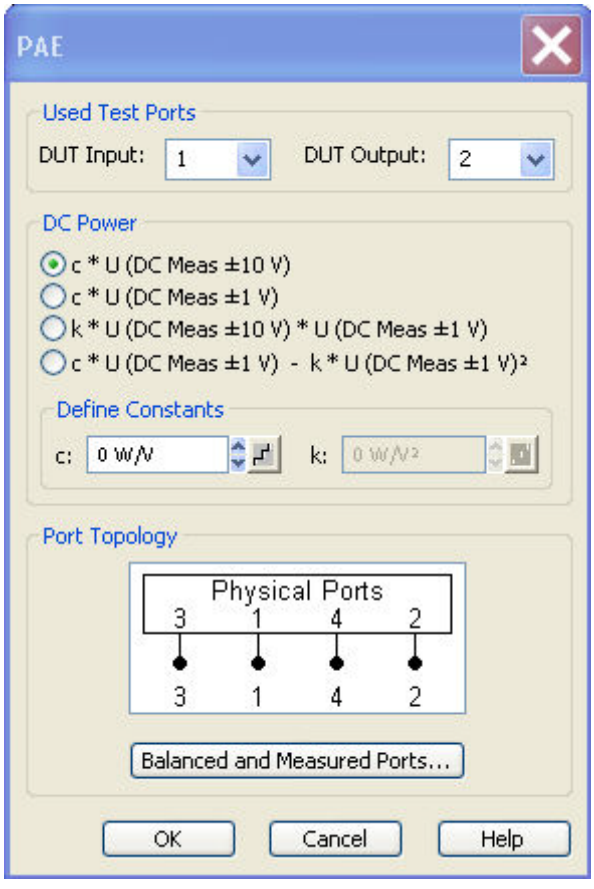
Create new trace and select name and measurement parameter:

```
CALCulate<Ch>:PARameter:SDEFine "<Trace_Name>", "DC+1V" |
"DC+10V"
```

## PAE...

Opens a dialog to select the Power Added Efficiency (PAE) of an active 2-port device as measured

quantity and to define the parameters for the PAE measurement.



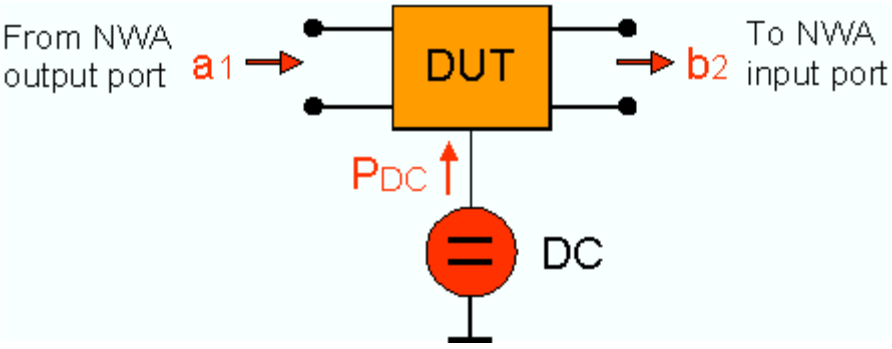

The *PAE* dialog provides the following settings:

- *Used Test Ports* selects the analyzer port providing the input signal $a_1$ *(DUT Input)* and the receiver port for the output signal $b_2$ *(DUT Output)*.
- The radio buttons and input fields in the *DC Power* panel select the test model and the parameters for measuring the DC power $P_{DC}$ supplied to the DUT. The constants can be incremented with the step size defined by clicking the buttons.
- *Select Topology* opens the Port Configuration dialog to define the properties of the test ports. Either physical test ports or logical ports can be used as *Input* and *Output* instrument ports. The graphics and messages displayed in the *Topology* panel are identical to the ones shown in the *More S-Parameters* dialog. When the unbalance-balance conversion is switched on, the PAE is calculated from the single-ended and differential mode wave quantities (see also *Mixed Mode Stability Factors)*.

*To obtain reasonable results, the test model and the respective Constant must be selected in accordance to the test setup.*

**Definition and measurement of PAE**

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.




**Definition:**

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 $b_2$ at the output of the DUT and the power of the incident wave $a_1$ at the input of the DUT; hence:

$$PAE = \frac{|b_2|^2 - |a_1|^2}{P_{DC}}$$

Positive PAE values indicate a gain in the RF power, negative values an attenuation. The PAE is always smaller than 1.

*The PAE is also available in true differential mode. The analyzer uses the differential wave quantities ad and bd to calculate the PAE.*

**Remote control:**

```
CALCulate<Ch>:PARameter:MEASure "<Trace_Name>", "PAE21" |
"PAE12" |...
```

Create new trace and select name and measurement parameter:

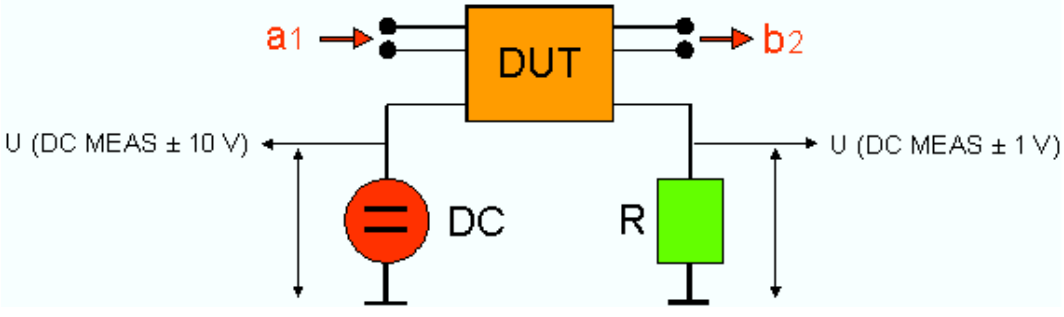
```
CALCulate<Ch>:PARameter:SDEFine "<Trace_Name>", "PAE21" |
"PAE12" |...
```

## DC Power Measurement

The power $P_{DC}$ supplied to the DUT can be measured using either one of the DC inputs *DC MEAS ± 10V* (for large voltages), *DC MEAS ± 1V* (for small voltages) or both inputs. The *DC Power* panel in the *PAE* dialog suggests different models involving different test setups and approximations. The models are selected by means of the radio buttons in the *DC Power* panel; they depend on the properties of the DC power supply (constant current $I_{DC}$ or constant power $U_{DC}$) and an optional precision resistor R used to measure the DC current. The values $I_{DC}$, $U_{DC}$, and R determine the *Constants* c and k. These constants must be entered in the *DC Power* panel, using the appropriate physical units, before a particular model can be activated.

- $I_{DC}$ = const., R = 0 —> $P_{DC}$ = c * U (DC Meas ±10 V) Assume that the DC power supply provides a constant current $I_{DC}$ and that the voltage applied to the DUT is measured via DC MEAS ± 10 V (R = 0). $P_{DC} = I_{DC} * U$(DC MEAS ± 10 V), hence the *Constant* c must be set equal to $I_{DC}$. The unit field shows the SI unit of a current (W/V). The input field for k is disabled (k is not used).
- $U_{DC}$ = const., R << $R_{DUT}$, $P_R$ = 0 —> $P_{DC}$ = c * U (DC MEAS ± 1 V) Assume that the DC power supply provides a constant voltage $U_{DC}$ and that the current through the DUT is measured by means of a precision resistor R connected in series and DC INP 2. If R << $R_{DUT}$ the power consumption of the resistor can be neglected so that $P_{DC} = U_{DC} / R * U$(DC MEAS ± 1 V), hence the *Constant* c must be set equal to $U_{DC} / R$. The unit field shows the SI unit of a current (W/V). The input field for k is disabled (k is not used).

- $R << R_{DUT}$, $P_R = 0$ —> $P_{DC}$ = k * U (DC MEAS ± 10 V) * U (DC MEAS ± 1 V) Assume that the DC power supply provides a constant voltage $U_{DC}$ and that the current through the DUT is measured by means of a precision resistor R connected in series and DC INP 2. $P_{DC} = U_{DUT} / R$ * U(DC MEAS ± 1 V) = $(U_{DC} - U_R)/ R$ * U(DC MEAS ± 1 V) = $U_{DC} / R$ * U(DC MEAS ± 1 V) – 1 / R * U(DC MEAS ± 1 V)², hence the *Constant* c must be set equal to $U_{DC} / R$ and k must be set equal to 1/R. The unit fields for c and k show the SI units of a current (W/V) and an inverse resistance (W/V²), respectively. Assume that the DC power supply provides an arbitrary (not necessarily constant) voltage $U_{DC}$ and current $I_{DC}$. The current through the DUT is measured by means of a precision resistor R connected in series and DC INP 2. The voltage applied to the DUT and R is measured via DC MEAS ± 10 V. If $R_{DUT}$ the power consumption of the resistor can be neglected so that $P_{DC}$ = 1/ R * U(DC MEAS ± 10 V) * U(DC MEAS ± 1 V), hence the *Constant k* must be set equal to 1/ R. The unit field shows the SI unit of an inverse resistance (W/V²). The input field for c is disabled (c is not used).
- $U_{DC}$ = const., $P_R = U (DC INP2)^2/R$ —> $P_{DC}$ = c * U (DC MEAS ± 1 V) + k * U (DC MEAS ± 1 V)^2 Assume that the DC power supply provides a constant voltage $U_{DC}$ and that the current through the DUT is measured by means of a precision resistor R connected in series and DC INP 2. $P_{DC} = U_{DUT} / R$ * U(DC MEAS ± 1 V) = $(U_{DC} - U_R)/ R$ * U(DC MEAS ± 1 V) = $U_{DC} / R$ * U(DC MEAS ± 1 V) – 1 / R * U(DC MEAS ± 1 V)², hence the *Constant* c must be set equal to $U_{DC} / R$ and k must be set equal to 1/R. The unit fields for c and k show the SI units of a current (W/V) and an inverse resistance (W/V²), respectively.




Models 1 and 4 provide the exact DC power supplied to the DUT, the two other models are based on the approximation $P_R = 0$.

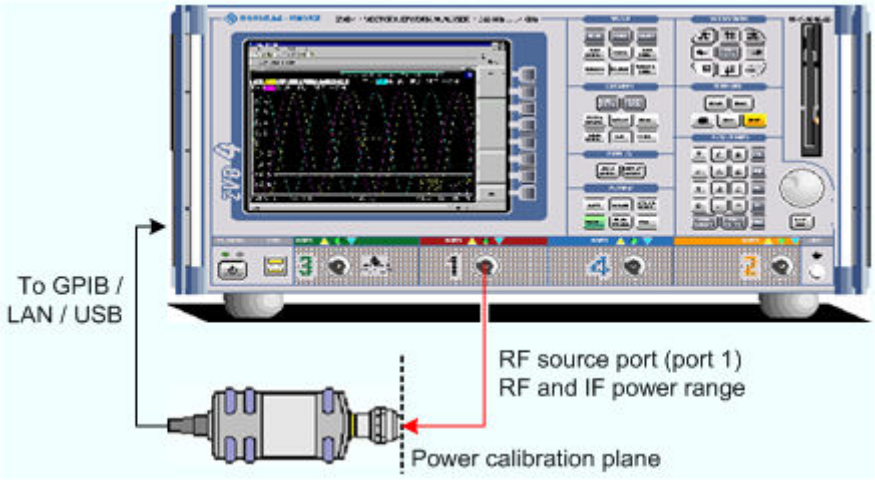
**Remote control:** `[SENSe<Chn>:]PAE:EXPRession`
`[SENSe<Chn>:]PAE:C`
`[SENSe<Chn>:]PAE:K`

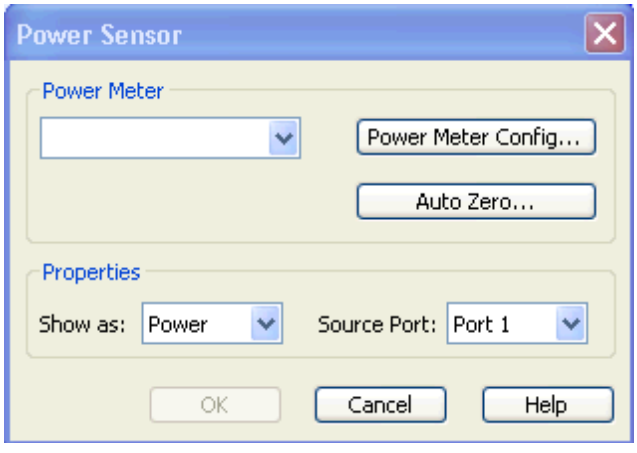
## Power Sensor...

Opens a configuration dialog for the measurement of wave quantities using an external power meter.

**Power Sensor measurement example**

The simplest test setup for a *Power Sensor* measurement involves one analyzer source port and a power sensor. The power sensor is connected e.g. to the analyzer's USB port and provides the (scalar) wave quantity results.




- The controls in the *Power Meter* panel show the configured power meters, open the *System Configuration – External Power Meters* dialog for adding and configuring external power meters, and start *Auto Zeroing* for the selected power meter.
- *Show as* selects the physical unit of the displayed trace. It is possible to display the measured *Voltage* U or convert the wave quantity into an effective power according to $P = V^2/Re(Z_0)$. $Z_0$ 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 *Port Configuration* dialog.
- *Source Port* selects one of the available test ports of the analyzer as a source of the stimulus signal.

**Remote control:**

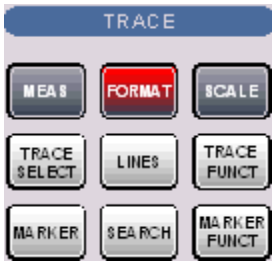
```
CALCulate<Ch>:PARameter:MEASure "<Trace_Name>", "Pmtr1D1" |
...
```

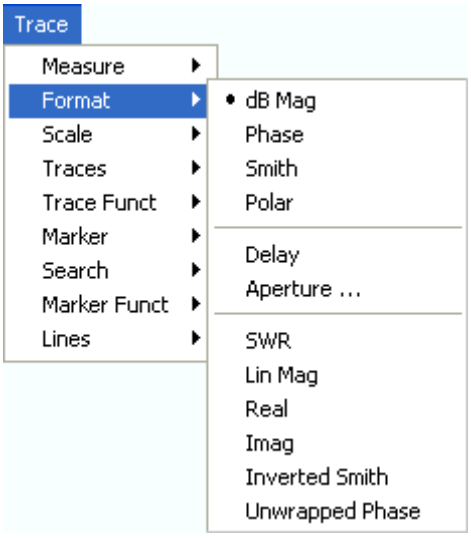
Create new trace and select name and measurement parameter:

```
CALCulate<Ch>:PARameter:SDEFine "<Trace_Name>", "Pmtr1D1"
|...
```

## Format

The *Format* submenu defines how the measured data is presented in the graphical display.




- *dB Mag* selects a Cartesian diagram with a logarithmic scale of the vertical axis to display the magnitude of a complex measured quantity.
- *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.
- *Smith* selects a Smith diagram to display an S-parameter or ratio.
- *Polar* selects a polar diagram to display an S-parameter or ratio.
- *Delay* calculates the group delay from an S-parameter or ratio and displays it in a Cartesian diagram.
- *Aperture* sets a delay aperture for the delay calculation.
- *SWR* calculates the Standing Wave Ratio from the measured reflection S-parameters and displays it in a Cartesian diagram.
- *Lin Mag* selects a Cartesian diagram with a linear scale of the vertical axis to display the magnitude of the measured quantity.
- *Real* selects a Cartesian diagram to display the real part of a complex measured quantity.
- *Imag* selects a Cartesian diagram to display the imaginary part of a complex measured quantity.
- *Inverted Smith* selects an inverted Smith diagram to display an S-parameter or ratio.
- *Unwrapped Phase* selects a Cartesian diagram with a linear vertical axis to display the phase of the measured quantity in an arbitrary phase range.

The *Format* settings are closely related to the settings in the *Scale* submenu and in the *Display* menu. All of them have an influence on the way the analyzer presents data on the screen.

*The analyzer allows arbitrary combinations of display formats and measured quantities (Trace – Measure). Nevertheless, in order 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; see Measured Quantities and Display Formats.*

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. Marker and trace formats can be applied independently.

## dB Mag

Selects a Cartesian diagram with a logarithmic 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.).

**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 control:** `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. $\phi$ (C) = arctan ( Im(C) / Re(C) ), appears on the vertical axis. $\phi$ (C) is measured relative to the phase at the start of the sweep (reference phase = 0°). If $\phi$ (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 *Phase Unwrapped* format avoids this behavior.

**Application:** Phase measurements, e.g. phase distortion, deviation from linearity.

**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 control:** `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 ($S_{11}$, $S_{22}$ etc.), 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 application example.

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 means of the *Marker Format* functions.

**Remote control:** `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 application example.

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 means of the *Marker Format* functions.

**Remote control:** `CALCulate<Chn>:FORMat POLar`

## Delay

Calculates the (group) delay from the measured quantity (primarily: from a transmission S-parameter) and displays it in a Cartesian diagram.

**Properties:** The group delay $\tau_g$ represents the propagation time of wave through a device. $\tau_g$ 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).

**Mathematical relations: Delay, Aperture, Electrical Length**

The group delay is defined as:

$$\tau_g = -\frac{d\phi_{rad}}{d\omega} = -\frac{d\phi_{deg}}{360°df}$$

where

$\phi_{rad/deg}$ = Phase response in radians or degrees

$\omega$ = Frequency/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 $\Delta$f and determining the corresponding phase change $\Delta\phi$. The delay is thus computed as:

$$\tau_{g,meas} = -\frac{\Delta\phi_{deg}}{360°\Delta f}$$

The aperture Δf must be adjusted to the conditions of the measurement.

If the delay is constant over the considered frequency range (non-dispersive DUT, e.g. a cable), then $\tau_g$ 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 of $L_{mech}$ *sqrt(ε)* 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).

**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.

The cables connecting the analyzer test ports to 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 mathematically 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.

**Remote control:** `CALCulate<Chn>:FORMat DELay`

## Aperture

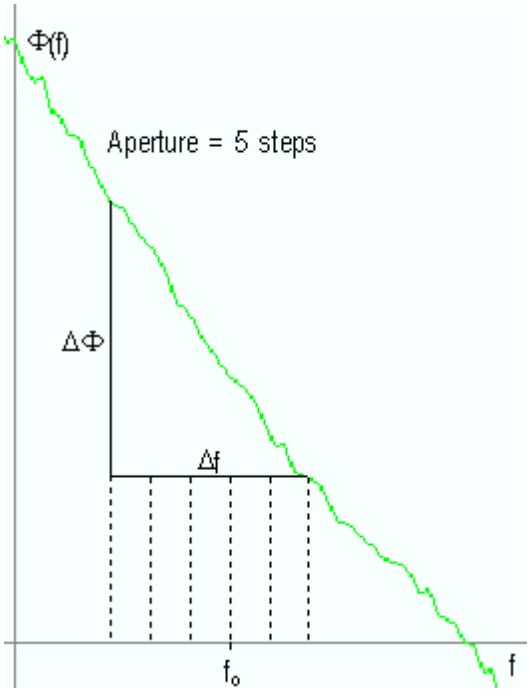
Sets a sets a delay aperture for the delay calculation. The aperture Δf is entered as an integer number of *Aperture Steps:*

The analyzer calculates the aperture from the sweep points of the current frequency sweep.

**Properties:** The delay at each sweep point is computed as:

$$\tau_{g,meas} = -\frac{\Delta\phi_{deg}}{360°\Delta f}$$

where the aperture Δf is a finite frequency interval around the sweep point $f_o$ and the analyzer measures the corresponding phase change Δϕ.




**Calculation of Δf and Δϕ**

With a given number of aperture steps n the delay at sweep point no. m is calculated as follows:

- If n is even (n = 2k), then Δf (m) = f (n+k) – f (n–k) and Δϕ(m)=Δϕ (n+k) – Δϕ (n–k).
- If n is odd (n = 2k+1), then Δf (m) = f (n+k) – f (n–k–1) and Δϕ(m)=Δϕ (n+k) – Δϕ (n–k–1).

It may be noted that the delay calculation is based on the already measured sweep points and does not slow down the measurement.

Δf is constant over the entire sweep range, if the sweep type is a Lin. Frequency sweep. For Log. Frequency and Segmented Frequency sweeps, it varies with the sweep point number m.

**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 noise effects, but at the expense of frequency resolution. Phase perturbations which are narrower in frequency than the aperture tend to be smeared over and can not be measured.

## SWR

Calculates the Standing Wave Ratio (SWR) from the measured quantity (primarily: from 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 the incident and the 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 control:** `CALCulate<Chn>:FORMat SWR`

## Lin Mag

*S*elects 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, DC Input 1/2, and the PAE) are always displayed in a Lin Mag diagram.

**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 control:** `CALCulate<Chn>:FORMat MLINear`

## Real

*S*elects 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.

**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 control:** `CALCulate<Chn>:FORMat REAL`

## Imag

*S*elects 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.

**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 control:** `CALCulate<Chn>:FORMat IMAGinary`

## Inverted Smith

*S*elects 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 ($S_{11}$, $S_{22}$ etc.), 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 application example.

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 means of the *Marker Format* functions.

**Remote control:** `CALCulate<Chn>:FORMat ISMith`

## Unwrapped Phase

*S*elects 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. $\phi$ (C) = arctan ( Im(C) / Re(C) ), appears on the vertical axis. $\phi$ (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 avoids artificial jumps of the trace but can entail a relatively wide phase range if the sweep span is large.

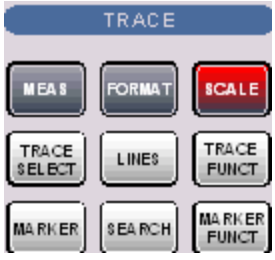
**Application:** Phase measurements, e.g. phase distortion, deviation from linearity.

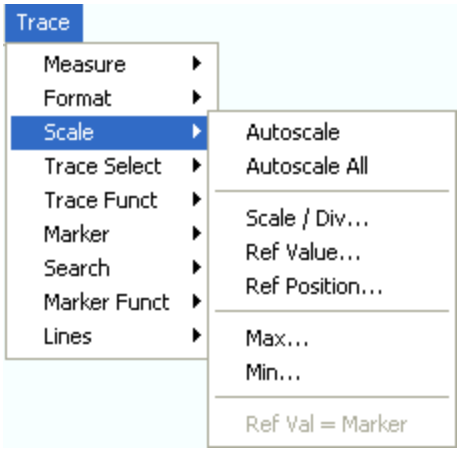
After changing to the *Unwrapped Phase* format, use *Trace – Scale – Autoscale* to re-scale the vertical axis and view the entire trace.

**Remote control:** `CALCulate<Chn>:FORMat UPHase`

## Scale

The *Scale* settings define how the current trace is presented in the diagram selected in the *Format* submenu.




- *Autoscale* adjusts the diagram scale in order to display the entire active trace in the diagram area.
- *Autoscale All* adjusts the diagram scale in order to display all traces in the diagram area.
- *Scale/Div.* sets the value of the vertical diagram divisions.
- *Ref. Value* sets the reference line of a Cartesian diagram or the outer circumference of a polar diagram.
- *Ref. Position* defines the position of the reference line in a Cartesian diagram.
- *Max* and *Min* define the upper and lower edge of a Cartesian diagram.
- *Ref Val = Marker* sets the reference value equal to the active marker value.

The *Scale* settings are closely related to the settings in the *Format* submenu and in the *Display* menu. All of them have an influence on the way the analyzer presents data on the screen.

The *Scale* settings depend on the diagram type (Trace – Format) 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. Position*, 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 disabled (grayed).

**Relations between the scaling parameters**

The scaling parameters *Scale / Div, Ref Value, Ref Position, Max, Min* are coupled together in the following manner:

Max – Min = Scale / Div * <Number of graticule divisions>
Max = Ref Value when Ref Position is 10
Min = Ref Value when Ref Position is 0

The Marker Functions provide a convenient alternative to manual diagram scaling.

## Autoscale

Adjusts the *Scale Divisions* and the *Ref. Value* in order to display the entire active trace in the diagram area, leaving an appropriate display margin.

- In Cartesian diagrams, the analyzer re-calculates the values of the vertical divisions so that the trace fits onto 80% of the vertical grid. The reference value is chosen to center the trace in the diagram.
- In circular diagrams *(Polar, Smith, Inverted Smith)*, the analyzer re-calculates 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.

Autoscale does not affect the stimulus values and the horizontal axis.

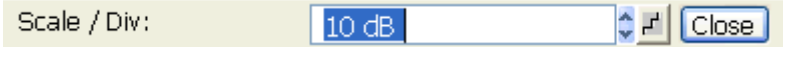
**Remote control:** `DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:Y[:SCALe]:AUTO ONCE`

## Autoscale All

Adjusts the *Scale Divisions* and the *Ref. Value* in order to display all traces in the diagram area, leaving an appropriate display margin. This scale settings are analogous to the *Autoscale* function. The traces in the active diagram area are taken into account irrespective of their channel assignment.

## Scale Div.

Sets the value of the vertical diagram divisions in Cartesian diagrams.



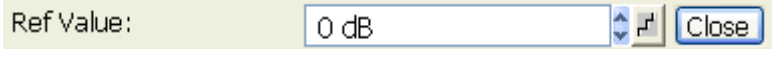

*Scale /Div* corresponds to the increment between two consecutive grid lines. The unit depends on the display format: dB for display format *dB Mag,* degrees for *Phase* and *Unwrapped Phase,* ns for *Delay,* U (units) for all other (dimensionless) formats.

*Scale /Div* is not available (grayed) for circular diagrams *(Polar, Smith, Inverted Smith)*.

**Remote control:** `DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:Y:PDIVision`

## Ref. Value

Sets the reference line of a Cartesian diagram or the outer circumference of a circular diagram.



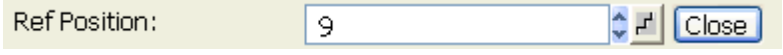

- In Cartesian diagrams *Ref. Value* defines the value of the reference line, indicated by a symbol at the right edge of the diagram area. The color of the symbol corresponds to the trace color. As the *Ref. Value* is varied, the position of the reference line (Ref. Position) is left unchanged, so that 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 *Unwrapped Phase,* ns for *Delay,* U (units) for all other (dimensionless) formats.
- In circular diagrams *(Polar, Smith, Inverted 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.

Use the paste marker list for convenient entry of the reference value.

**Remote control:** `DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:Y:RLEVel`

## Ref. Position

Defines the position of the reference line in a Cartesian diagram.




The reference line is indicated by a symbol at the right edge of the diagram area. The color of the symbol corresponds to the trace color. *Ref. Position* is defined on a linear scale between 0 (bottom line of the diagram) and 10 (top line of the diagram). As the *Ref. Position* is varied, the value of the reference line (Ref. Value) is left unchanged, so the current trace is shifted together with the *Ref. Position*.

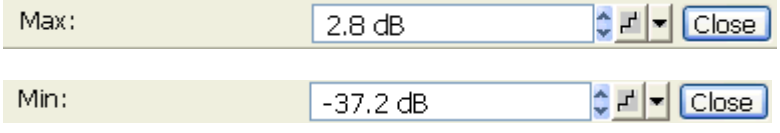
*Ref. Position* is not available (grayed) for polar diagrams *(Polar, Smith, Inverted Smith)*.

Use the analyzer's drag-and-drop functionality to move the reference line symbol to the desired position.

**Remote control:** `DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:Y:RPOSition`

## Max and Min

Define the upper and lower edge of a Cartesian diagram.




*Max* and *Min* are not available (grayed) for polar diagrams *(Polar, Smith, Inverted Smith)*.

**Remote control:** `DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:Y:TOP`
`DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:Y:BOTTom`

## 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.

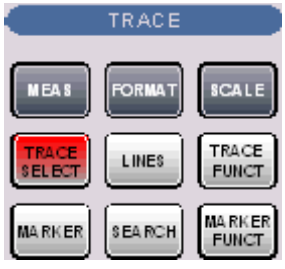
- In Cartesian diagrams the operation shifts the trace in vertical direction.
- In polar diagrams *(Polar, Smith, Inverted Smith),* the operation re-defines the value of the outer circumference and therefore enlarges or scales down the diagram, leaving the center unchanged.

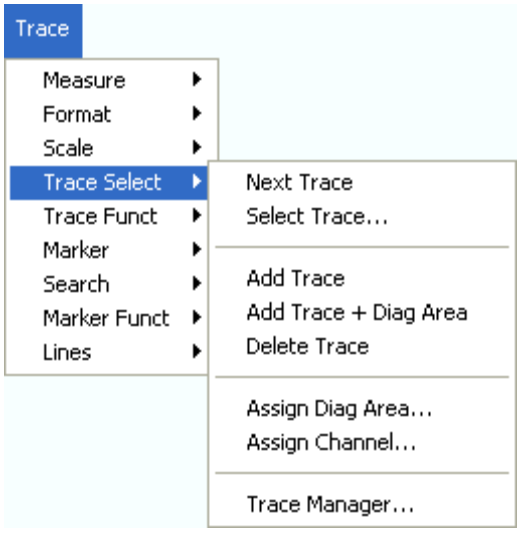
Use the analyzer's drag-and-drop functionality to move the reference line symbol to the desired position.

**Remote control:** No command, display configuration only

## Trace Select

The *Trace Select* submenu provides functions to handle traces and diagram areas, and assign traces to channels.



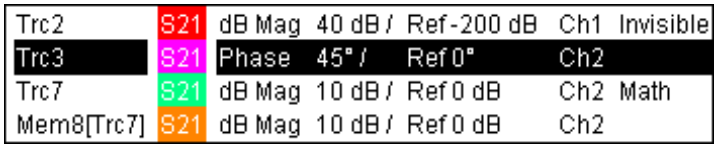




- *Next Trace* selects the next trace as the active trace (disabled if only one trace is defined).
- *Select Trace* opens a box to select an arbitrary trace of the active setup as the active trace (disabled if only one trace is defined).
- *Add Trace* creates a new trace in the current diagram area.
- *Add Trace + Diag. Area* generates a new trace in a new diagram area.
- *Delete Trace* deletes the active trace.
- *Assign Channel* assigns the active trace to another channel.
- *Assign Diag. Area* assigns the active trace to another diagram area.
- *Trace Manager* opens a dialog to perform the previous actions systematically for all traces and diagram areas.

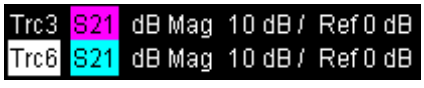
**Active and inactive traces**

The screen can display several diagram areas simultaneously, each with a variable number of traces.

- In an active diagram area one of these traces is the active trace. The active trace is highlighted in the trace list on top of the diagram area (Trc 3 in the figure below):




- If an inactive area is selected as the active area, the trace that was active last time when the area was active will again become the active trace. It is highlighted in the trace list of the inactive diagram area as shown for Trc6 in the figure below:

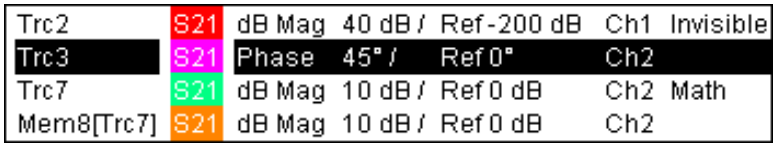



All settings in the *Trace* menu except the *Trace Functions* involving a memory trace apply to the active trace in the active diagram area.

*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 Active Traces in Remote Control.*

## Next Trace

Selects the next trace in a series of displayed traces as the active trace. This function is disabled if only one trace is defined.



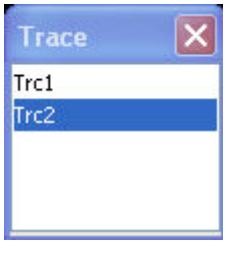

*Next Trace* has the following meaning:

- In a list of traces displayed in a common diagram area, the next trace is the trace below the current trace (in the figure above, Trc 3 is the active trace and Trc 4 is the next trace).
- If the active trace is the last trace in the list, the next trace is the first trace in the trace list of the next diagram area.

**Remote control:** The numeric suffix `<Ch / Tr>` appended to the first-level mnemonic of a command selects a trace as active trace.

## Select Trace

Opens a box to select an arbitrary trace of the active setup as the active trace. This function is disabled if only one trace is defined.



**Remote control:** The numeric suffix `<Ch / Tr>` appended to the first-level mnemonic of a command selects a trace as active trace.

## Add Trace

Creates a new trace in the current diagram area and assigns it to the current channel. The new trace is created with the trace and channel settings of the former active trace but displayed with another color. The former and the new active trace are superimposed but can be easily separated, e.g. by changing the *Reference Position.*

The new trace is named Trc <n>, where <n> is the largest of all existing trace numbers plus one. The name can be changed in the *Trace Manager*.

To create a new trace in a new channel, use *Channel – Channel Select – New Channel.*

**Remote control:**
```
CALCulate<Ch>:PARameter:SDEFine <Trace Name>, < Meas Parameter>
DISPlay:WINDow<Wnd>:TRACe<WndTr>:FEED
```

## Add Trace + Diag. Area

Creates a new trace in a new diagram area and assigns the trace to the current channel. The new trace is created with the trace and channel settings of the former active trace but displayed with another color.

The new trace is named Trc <n>, where <n> is the largest of all existing trace numbers plus one. The name can be changed in the *Trace Manager*.

**Remote control:**
```
CALCulate<Ch>:PARameter:SDEFine <Trace Name>, < Meas Parameter>
DISPlay:WINDow<Wnd>:STATe ON
DISPlay:WINDow<Wnd>:TRACe<WndTr>:FEED
```

## Delete Trace

Deletes the current trace and removes it from the diagram area. *Delete Trace* is disabled if the setup contains only one trace: In manual control, each setup must contain at least one diagram area with one channel and one trace.

To restore a trace that was unintentionally deleted, use *Undo*.

**Remote control:**
```
CALCulate<Ch>:PARameter:DELete <Trace Name>
```

## Assign Channel

Assigns the active trace to another channel. A popup window offers a list of all channels available:

- Selecting one of the existing channel names assigns the current trace to the existing channel.
- Selecting *New* creates a new channel and assigns the current trace to the new channel. 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.

*Assign Channel is disabled if the current setup contains only one channel. To create an additional channel, select Channel – Channel Select – New Channel.*

**Remote control:** `CALCulate<Ch>:PARameter:SDEFine <Trace Name>, < Meas Parameter>`

## Assign Diag. Area

Assigns the active trace to another diagram area. A popup window offers a list of all areas available:

- Selecting one of the existing area numbers assigns the active trace to the existing diagram area: The active trace is removed from the previous area and displayed in the new diagram area.
- Selecting *New* creates a new diagram area and assigns the active trace to the new area. The new area is numbered <n>, where <n> is the largest of all existing area numbers plus one.

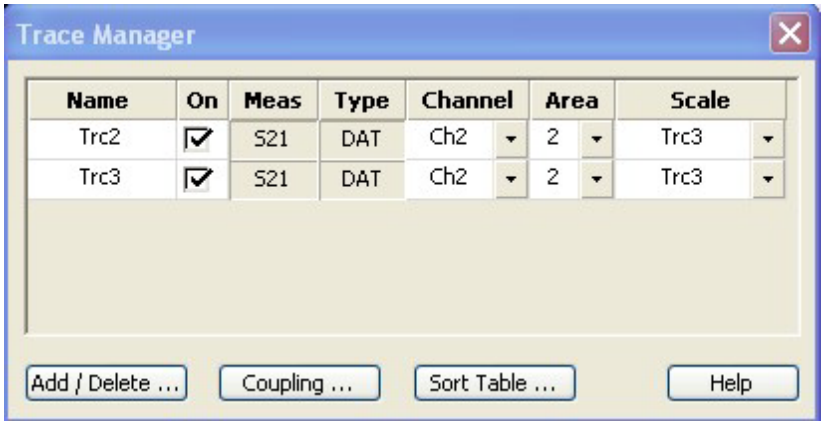
*Assign Diag. Area is disabled if the current setup contains only one area. To create an additional area, click Display – Area Select – New Diag. Area.*

To go to another diagram area and activate the last active trace in this area, simply click a point inside the new area.

**Remote control:** `DISPlay:WINDow<Wnd>:TRACe<WndTr>:FEED <Trace Name>`

## Trace Manager

Opens a dialog to perform the actions in the *Trace Select* menu systematically for all traces and diagram areas.




All existing traces of the current setup are listed in a table with several editable (white) or non-editable (gray) columns. Below the table the *Trace Manager* provides the following buttons:

- *Add/Delete...* opens a dialog to add a new trace or delete a trace.
- *Coupling* opens a dialog to define coupling criteria (channel, scale) for all traces in the table.
- *Sort Table* opens a dialog to change the order of the traces (rows) in the table.

**Columns in the *Trace Manager* table**

- *Name* indicates the current trace name. The default names for new traces are Trc<n> where <n> is a current number. Current numbers in the trace names are necessary to make automatic assignments, e.g. decouple the channel settings in the *Coupling* dialog. To serve as an unambiguous reference, trace names must be unique across all channels and diagram areas.
- *On* indicates whether the trace is displayed on the screen *(On)* or invisible.
- *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 the channel of each trace.
- *Area* indicates the diagram area of each trace.
- *Scale* shows which traces use common scaling and format settings.

**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. This places some restrictions on the syntax of trace names:

- The first character of a trace name can be either one of the upper case letters A to Z, one of the lower case letters a to z, an underscore _ or a square bracket [ or ].
- For all other characters of a trace name, the numbers 0 to 9 can be used in addition.

The analyzer does not accept illegal trace names. If an illegal name is specified, the input field in the *Trace Manager* turns red.

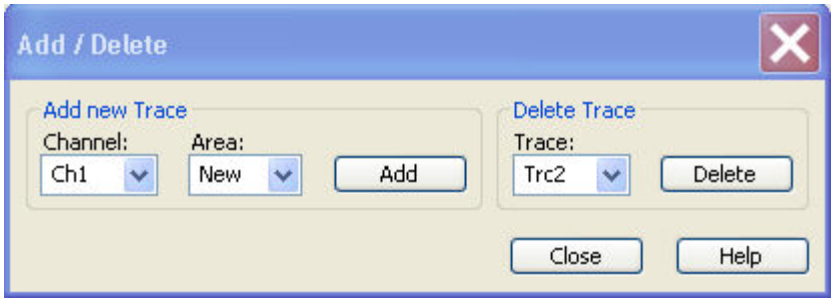
**Remote control:**

```
CALCulate<Ch>:PARameter:SDEFine <Trace Name>, < Meas Parameter>
CONFigure:CHANnel<Ch>:TRACe:REName
CONFigure:TRACe<Trc>:CATalog?
CONFigure:TRACe<Trc>:NAME
```

```
CONFigure:TRACe<Trc>:NAME:ID?
CONFigure:TRACe:REName
```

## Add/Delete

Opens a dialog to add a new trace or delete a trace.



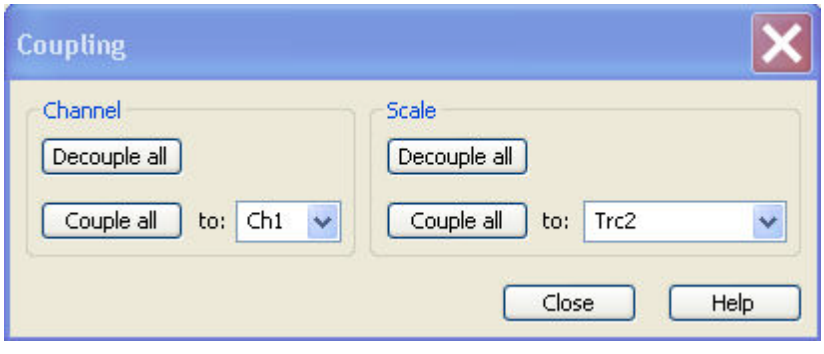

- *Add* creates a new trace and adds it to the list in the *Trace Manager,* assigning it to the channel and diagram area selected in the drop-down lists. It is possible to create a *New* channel and/or diagram area for the new trace.
- *Delete* deletes the selected trace, removing it from the list in the *Trace Manager* and from the screen. This button is disabled if the setup contains only one trace: In manual control, each setup must contain at least one diagram area with one channel and one trace.

**Remote control:**

```
CALCulate<Ch>:PARameter:SDEFine <Trace Name>, < Meas
Parameter>
DISPlay:WINDow<Wnd>:TRACe<WndTr>:FEED <Trace Name>
DISPlay[:WINDow<Wnd>]:TRACe:EFEed <Trace Name>
CALCulate<Ch>:PARameter:DELete <Trace Name>
```

## Coupling

Selects common channel or scale settings for all traces in the *Trace Manager* dialog.




The channel and scale coupling is set in two independent panels.

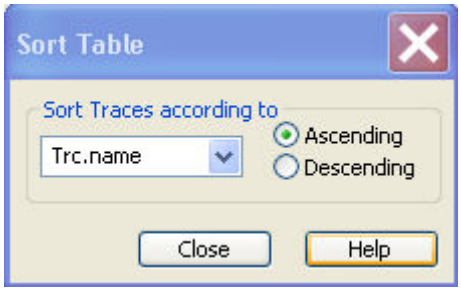
- *Decouple All* assigns independent channel or scale settings to all traces in the *Trace Manager*. If the channel and trace names include numbers, the trace with the lowest number is assigned to the channel with the lowest number and so forth. Measurement or data traces and their associated memory traces are assigned to the same channel.
- *Couple All* assigns all traces to the channel or scale settings selected in the corresponding

drop-down lists. All channel or scale settings except the selected ones are lost. The analyzer displays a confirmation dialog box before deleting the unused channels.

**Remote control:** –

### Sort Table

Changes the order of the traces (table rows) in the *Trace Manager* dialog.



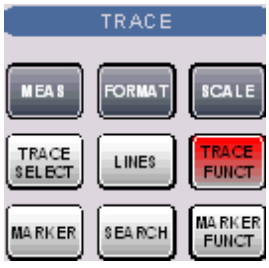

- The drop-down list contains all trace properties that can provide the sorting criterion. The properties correspond to the column headers in the *Trace Manager.*
- The two radio buttons qualify whether the traces in the table are sorted in ascending or descending order, according to the property selected in the pull-down list.

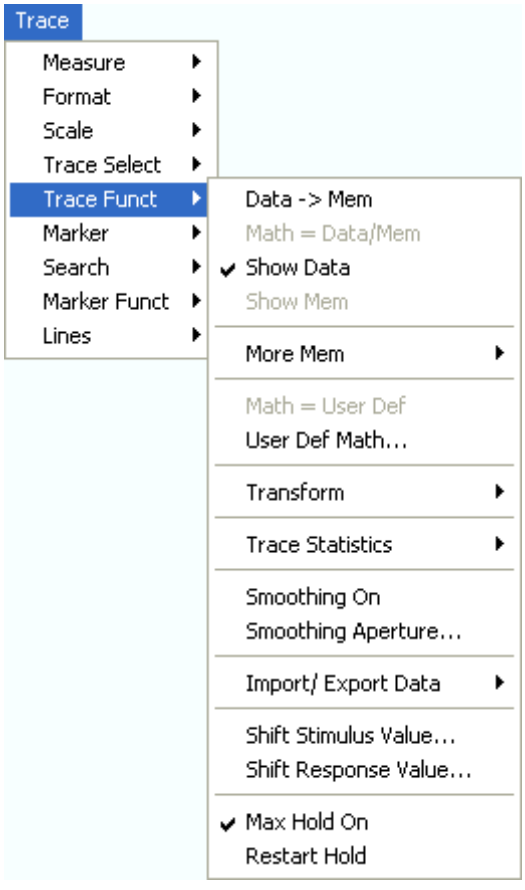
**Remote control:** No command, display configuration only.

## Trace Funct

The *Trace Funct(ions)* store traces to the memory and perform mathematical operations on traces

Refer to section *Trace Types* in the *System Overview* chapter to learn more about data traces, memory traces, and mathematical traces.




- *Data -> Mem* stores the active data trace as a memory trace.
- *Math = Data/Mem* activates the mathematical mode where the data trace is divided by the memory trace.
- *Show Data* shows or hides the active data trace.
- *Show Mem* shows or hides the active memory trace.
- *More Mem* opens a submenu to store traces as memory traces, show or hide traces.
- *Math = User Def* activates the mathematical mode with a user-defined relation between traces.
- *User Def Math...* opens a dialog to define a mathematical relation between traces.
- *Transform* opens a submenu with the time domain transformation functions (with option ZVAB-K2, *Time Domain).*
- *Trace Statistics* opens a submenu to evaluate statistical and phase information of the trace.
- *Smoothing On* activates the smoothing function for the active trace.
- *Smoothing Aperture...* defines how many measurement points are averaged to smooth the trace.
- *Import/Export Data* stores the active trace to a file or loads a memory trace from a file.
- *Shift Response Value*... opens a dialog to define a user-correction to the measured values.
- *Shift Stimulus Value...* shifts a memory trace in horizontal direction.
- *Max Hold On* enables or disables the max hold (peak hold) function for the active trace.
- *Restart Hold* restarts the max hold (peak hold) function for the active trace, discarding the old maximum measurement results.

Many of the functions of the *Trace Funct* menu act on the active trace. Data traces and the associated memory traces share many of their properties; see coupling of memory traces.

## Data -> Mem

Stores the current state of the active trace as a memory trace. The memory trace is displayed in the active diagram area with another color, and its properties are indicated in the trace list:

Memory traces are named *Mem<n>[<Data_Trace>]* where <n> counts all data and memory traces in the active setup in chronological order, and <Data_Trace> is the name of the associated data trace. Trace names can be changed in the *Trace Manager* dialog.

The exact function of *Data -> Mem* depends on the number of memory traces associated to the active data trace:

- If no memory trace is associated with the active trace, a new memory trace is generated.
- If several memory traces are associated with the active trace, the current measurement data overwrites the last generated or changed memory trace.

To store the current measurement data to a new memory trace (without overwriting an existing memory trace), or select and overwrite a particular memory trace, use the *-> Mem* dialog. You can also create multiple memory traces using the *Import Data* dialog. Notice that it is not possible to store *Max. Hold* traces to memory.

**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 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 settings in the *Trace – Format* menu
- All settings in the *Trace – Scale* menu

Selection of the measured quantity *(Trace – Measure)* 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 the sweep type of a data trace is changed so that the stimulus ranges of the data traces and the memory traces become incompatible, all coupled memory traces are removed from the diagram area and deleted.

**Remote control:** `CALCulate<Chn>:MATH:MEMorize`

## Math = Data/Mem

Activates the mathematical mode where the active data trace is divided by the last generated memory trace. The division is calculated on a point-to-point basis: Each measurement point of the active trace is divided by the corresponding measurement point of the memory trace. 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 Data/Mem is well-defined.

*Math = Data/Mem* and *Math = User Def* are alternative options of the mathematical mode. Selecting one option disables the other. The mathematical expression defined via User Def Math is not affected.

**Remote control:**
```
CALCulate<Chn>:MATH[:EXPRession]:SDEFine <string>
CALCulate<Chn>:MATH:STATe ON
```

## Show Data

Displays or hides the active data trace in the diagram area. If one of the mathematical options *Math = Data/Mem* or *Math = User Def* are active, then the active mathematical trace is displayed or hidden.

**Remote control:** No command, display configuration only.

## Show Mem

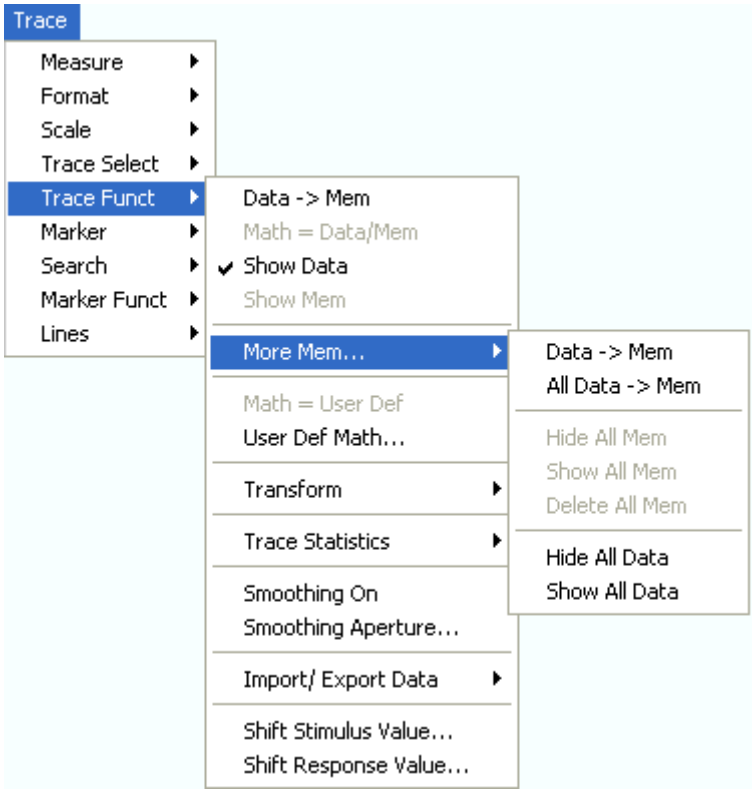
Displays or hides the active memory trace in the diagram area or the memory trace associated with the active data trace.

- If no memory trace is associated with the active data trace, *Show Mem* is disabled.
- If several memory traces are associated with the active data trace (see *-> Mem)*, *Show Mem* affects the last generated or changed memory trace.

**Remote control:** No command, display configuration only.

## More Mem

Opens a submenu to store traces as memory traces, show or hide traces. The submenu complements the *Data -> Mem* menu command.

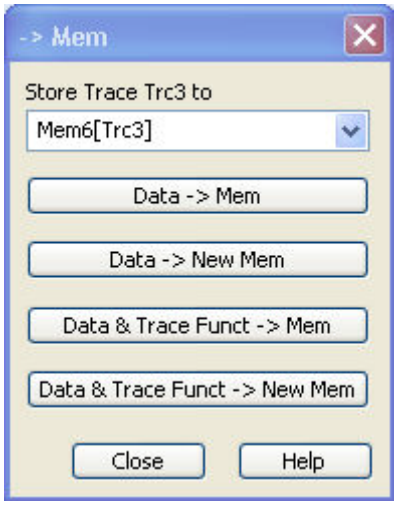



- *Data —> Mem* opens a dialog to store the active trace to a memory trace.
- *All Data —> Mem* opens a dialog to store all traces in the active setup to memory traces.
- The remaining functions hide or show all data traces or memory traces. Moreover it is possible to delete all memory traces in the active setup.

**Remote control:** `TRACe:COPY <memory_trc>,<data_trc>`

## -> Mem Dialog

The *Data -> Mem* dialog stores the active trace a memory trace.




- *Store Trace <trace_name> to* contains all memory traces associated with the active data trace.

The selected memory trace can be replaced.

- *Data -> Mem* stores the active data trace to the memory trace selected above. If no memory trace is associated with the current trace, then a new memory trace is created. The new trace is named *Mem<n+1>[<Data_Trace>],* where n is the largest of all existing memory trace indices.
- D*ata -> New Mem* stores the active data trace to a new memory trace, leaving the existing memory traces unchanged.
- *Data & Trace Funct. -> Mem* stores the current state of the active data trace modified by the trace functions to the memory trace selected above. If no memory trace is associated with the current trace, then a new memory trace is created.
- *Data & Trace Funct. -> New Mem* stores the current state of the active data trace modified by the trace functions to a new memory trace, leaving the existing memory traces unchanged.

**Trace functions**

The trace functions comprise the following mathematical operations:

- Any mathematical relation applied to the trace *(Math = Data/Mem, Math = User Def)*.
- A shift of the trace in horizontal or vertical direction *(Shift Response Value, Shift Stimulus Value).*

*Data —> Mem* and *Data —> NewMem* store the raw trace without the trace functions, *Data & Trace Funct —> Mem* and *Data & Trace Funct —> NewMem* store the trace after it has been transformed using the trace functions.

**Remote control:** `TRACe:COPY <memory_trc>,<data_trc>`

## All Data -> Mem

The *All Data -> Mem* dialog stores all traces in the active setup to memory traces.

- *Data -> Mem* stores all data traces in the current setup to memory traces. If no memory trace is associated with a data trace, then a new memory trace is created. The new trace is named *Mem<n+1>[<Data_Trace>],* where n is the largest of all existing memory trace indices. If several memory traces are associated with a data trace, the newest memory trace is replaced.
- D*ata -> New Mem* stores all data traces in the current setup to memory traces, leaving the existing memory traces unchanged.
- *Data & Trace Funct. -> Mem* stores the current state of all data traces modified by the trace functions to memory traces. If a trace has no memory trace, then a new memory trace is

created.

- *Data & Trace Funct. -> New Mem* stores the current state of all data traces modified by the trace functions to new memory traces, leaving the existing memory traces unchanged.

**Trace functions**

The trace functions comprise the following mathematical operations:

- Any mathematical relation applied to the trace *(Math = Data/Mem, Math = User Def)*.
- A shift of the trace in horizontal or vertical direction *(Shift Response Value, Shift Stimulus Value).*

*Data —> Mem* and *Data —> NewMem* store the raw trace without the trace functions, *Data & Trace Funct —> Mem* and *Data & Trace Funct —> NewMem* store the trace after it has been transformed using the trace functions.

**Remote control:** `TRACe:COPY <memory_trc>,<data_trc>`

## Math = User Def

Activates the mathematical mode and displays the mathematical trace defined via User Def. Math. 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.

This function is disabled unless a user-defined mathematical trace is available.

*Math = User Def* and *Math = Data/Mem* are alternative options of the mathematical mode. Selecting one option disables the other. The mathematical expression defined via Define Math is not affected.

**Remote control:**
```
CALCulate<Chn>:MATH[:EXPRession]:SDEFine <string>
CALCulate<Chn>:MATH:STATe ON
```

## User Def. Math...

Opens a dialog to define a mathematical relation between traces and calculate a new mathematical trace. Each measurement point of the active trace is replaced by the corresponding point of the mathematical trace.

**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:

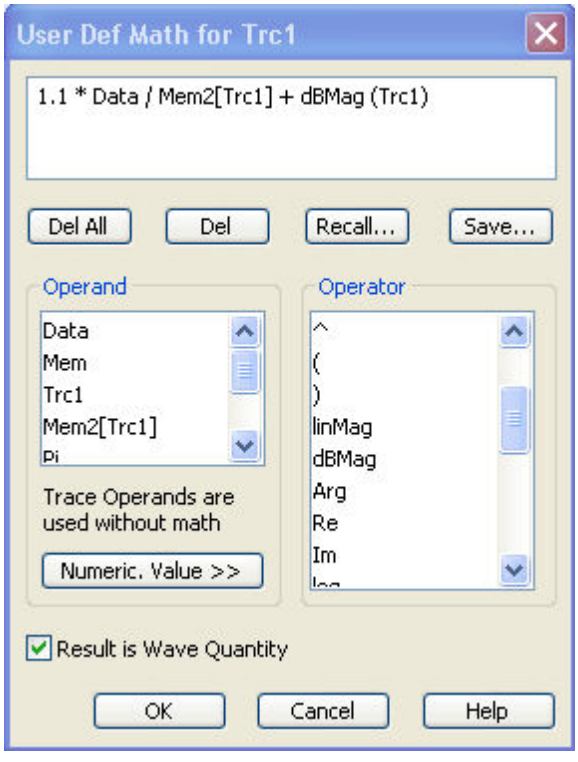
$$Math_i = Fct.\ (c_1, ..., c_n, Trc1_i, Trcm_i),\ i = 1, \text{no. of points}$$

Different traces can be used in the same mathematical relation provided that 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.*




In its upper part the dialog contains a window to view and edit the mathematical expression and four buttons:

- Del. All clears the window, so it is possible to start editing a new expression.
- Del. deletes the last character before the current cursor position (backspace function). Deleting a character of an operand causes the operand to become invalid.
- Recall... calls up an *Open File* dialog to load a mathematical expression from 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.
- Save... calls up a *Save As...* dialog to store the current expression 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.

To import a math file (*.mth) you can also use the Windows Explorer and simply double-click the file or drag and drop the file into the NWA application. You must enable the mathematical mode separately.

The operands and operators in the expression can be selected from two lists and the *Numeric Value* panel:

- Operand contains all data traces and memory traces of the active setup plus the constants pi (approx. 3.14159), e (approx. 2.71828) and the imaginary unit j. The trace operands denote unmodified data and memory traces. Mathematical relations for the traces are not taken into account, even if the mathematical mode is active *(Math = User Def)*. The same applies to smoothing and *Shift Response Value.*

- **Operator**

  Contains operators for arithmetic operations and mathematical functions. The following table lists how the operators act on a complex quantity z = x + jy.

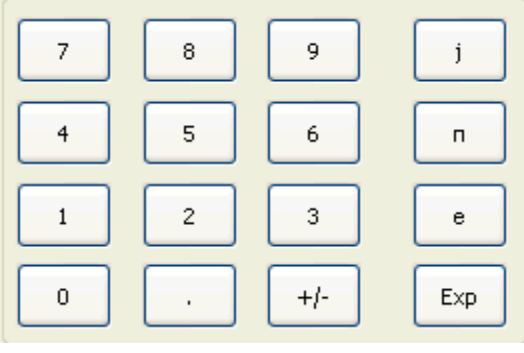
| | |
|---|---|
| +, -, *, / | Basic arithmetic operations |
| () | Grouping parts of an expression |
| linMag | \|z\| = sqrt ( $x^2$ + $y^2$ ) |
| dBMag | dB Mag(z) = 20 * log\|z\| dB |
| Arg | Phase $\phi$ (z) = arctan ( Im(z) / Re(z) ) |
| Re, Im | x, y (Real and Imag) |
| log, ln | Common (base 10) 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 function 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, which may be converted into a dB value (for *dB Mag* trace format).
  - In power sweeps, StimValprovides the voltage in V that results from the source power in dBm. To obtain the correct source power in dBm (for *dB Mag* trace format), *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 *Port Configuration* dialog.
  - In time sweeps, StimVal is the stimulus time in s.
  - In CW mode sweeps, StimVal is the number of the point.

- **Numeric Value**

  extends the *Define Math* dialog and opens the numeric input panel (toggle function):




  The numeric input panel supports the entry of numeric values and constants. 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
  - *Exp* defines a number in exponential representation, e.g. 12 exp 3 for 12*10^3 = 12000.

  Products of numbers and constants may be entered in abbreviated form, e.g. 2e for 2*e.

NOTE *If traces are used as operands, the unmodified, **linear** complex trace data (no dB-values) enter*

*into the mathematical expression, irrespective of the current trace format. No mathematical traces are available as operands.*

*Result is Wave Quantity* in the lower part of the dialog controls the conversion and formatting of the mathematic expression.

- 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 and numeric example**

In the *More Wave Quantities* dialog, the *Show as:* control element specifies whether wave quantities are displayed as voltages or equivalent powers, 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 (checked), 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.

See also example for `CALCulate<Chn>:MATH:WUNit:STATe ON | OFF`.

## Transform

Opens a submenu to view the measurement results as a function of time. The time domain transformation is enabled if option ZVAB-K2, *Time Domain,* is available.

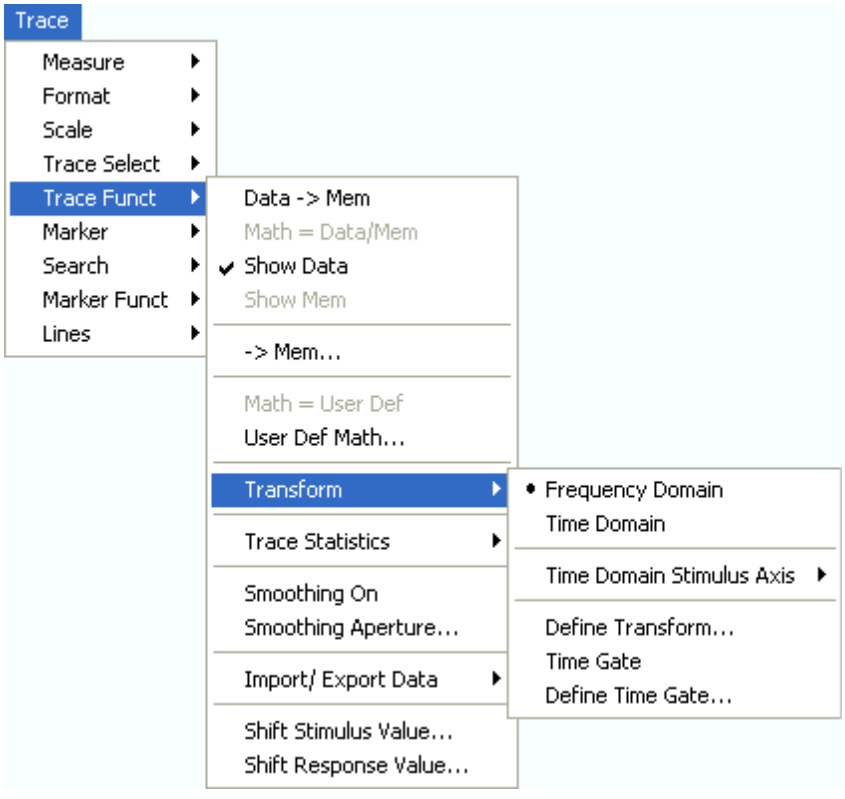
**Time domain transformation**

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 in order 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 sidelobes due to the finite sweep range. Moreover, it is possible to eliminate unwanted responses by means of 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 1EZ44_OE which is posted on the R&S internet.




- *Frequency Domain* and *Time Domain* select frequency domain or time domain representation of the active trace.
- *Time Domain Stimulus Axis* opens a submenu to define the stimulus axis range for the time domain representation.
- *Define Transform*... opens a dialog to define the transformation type and the frequency domain window used to optimize the time domain response.
- *Time Gate* switches the time gate defined with *Define Time Gate...* on or off.
- *Define Time Gate*... opens a dialog to select the time gate and define its parameters.

## Frequency Domain

Selects the frequency domain representation for the active trace. The softkey is enabled if option ZVAB-K2, *Time Domain,* is available, and if a linear frequency sweep *(Channel – Sweep – Sweep Type – Lin. Frequency)* is active.

In frequency domain representation the diagram shows the measured trace as a function of the stimulus frequency applied to the DUT. The trace corresponds to the results obtained during the frequency sweep, however, the effect of a time gate is taken into account as long as the *Time Gate* function is active. The x-axis corresponds to the sweep range (stimulus range) selected via *Channel – Stimulus*.

**Gated and ungated state of the frequency domain representation**

The trace in the frequency domain depends on the state of the *Time Gate:*

- If the gate is disabled, the frequency domain trace corresponds to the sweep results prior to the transformation.
- If the gate is enabled, the frequency domain trace shown is calculated from the time domain trace which is gated and transformed back into the frequency domain.

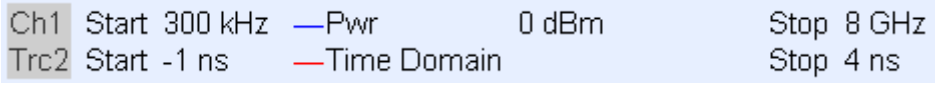
In time domain representation, you can use the *Time Gate* settings in order to eliminate unwanted responses in your signal. After switching back to the frequency domain, you will receive the frequency response of your DUT without the contribution of the unwanted responses.

**Remote control:** `CALCulate<Chn>:TRANsform:TIME:STATe OFF`

## Time Domain

Selects the time domain representation for the active diagram area. The softkey is enabled if option ZVAB-K2, *Time Domain,* is available, and if a linear frequency sweep *(Channel – Sweep – Sweep Type – Lin. Frequency)* is active. The analyzer automatically quits time domain representation as soon as a different sweep type is selected.

In time domain representation the diagram shows the measurement results as a function of time. The results are obtained by transforming the measured frequency sweep data into the time domain using an appropriate mathematical transformation type and frequency window *(Define Transform...)*. 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 settings in the *Time Domain Stimulus Axis* submenu 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.

**Properties of the 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:

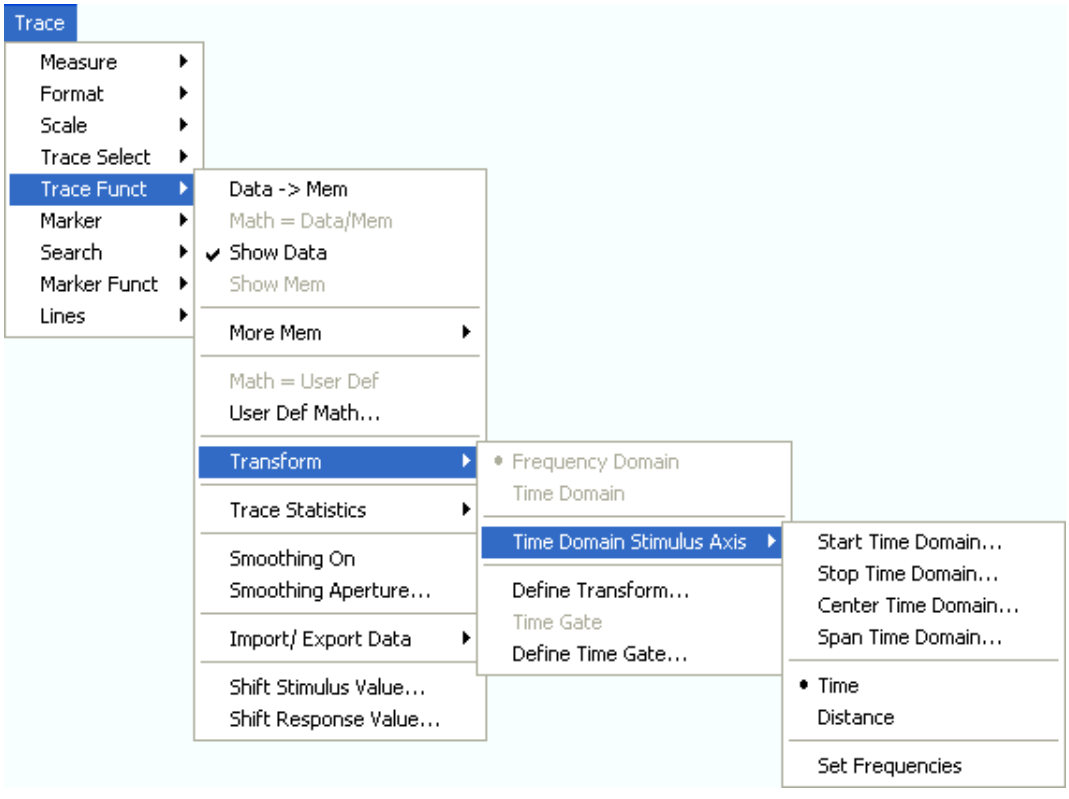
- The frequency points must be equidistant. If a *Log Frequency* or *Segmented Frequency* sweep is active, the measurement points are interpolated.
- The time domain response is repeated after a time interval which is equal to $\Delta t = 1/\Delta f$, where $\Delta f$ is the spacing between two consecutive sweep points in the frequency domain. For a sweep span of 4 GHz and 201 equidistant sweep points, $\Delta f = 4\ GHz/200 = 2 * 10^7$ Hz, so that $\Delta t = 50$ ns. $\Delta t$ is termed measurement range (in time domain) or unambiguous range.

Additional constraints apply if the selected Chirp z-transformation is a lowpass transformation.

**Remote control:** `CALCulate<Chn>:TRANsform:TIME:STATe ON`

## Time Domain Stimulus Axis

Opens a submenu to define the stimulus axis range for time domain representation.




- *Start Time Domain* is the lowest displayed time and corresponds to the left edge of the Cartesian diagram.
- *Stop Time Domain* is the highest displayed time and corresponds to the right edge of the Cartesian diagram.
- *Center Time Domain* corresponds to the center of the Cartesian diagram, i.e. (Start + Stop)/2.
- *Span Time Domain* corresponds to the diagram width, i.e. (Stop – Start).
- *Time* and *Distance* switch over between the x-axis scaling in time units or distance units.
- *Set Frequencies* opens the *Channel – Stimulus* menu in order to change the stimulus axis in the frequency domain.

Use the paste marker list for convenient entry of *Start* and *Stop* values.

**Distance units for transmission and reflection parameters**

The interpretation of time and distance depends on the measurement type. For reflection measurements, 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* calculation is consistent with this interpretation:

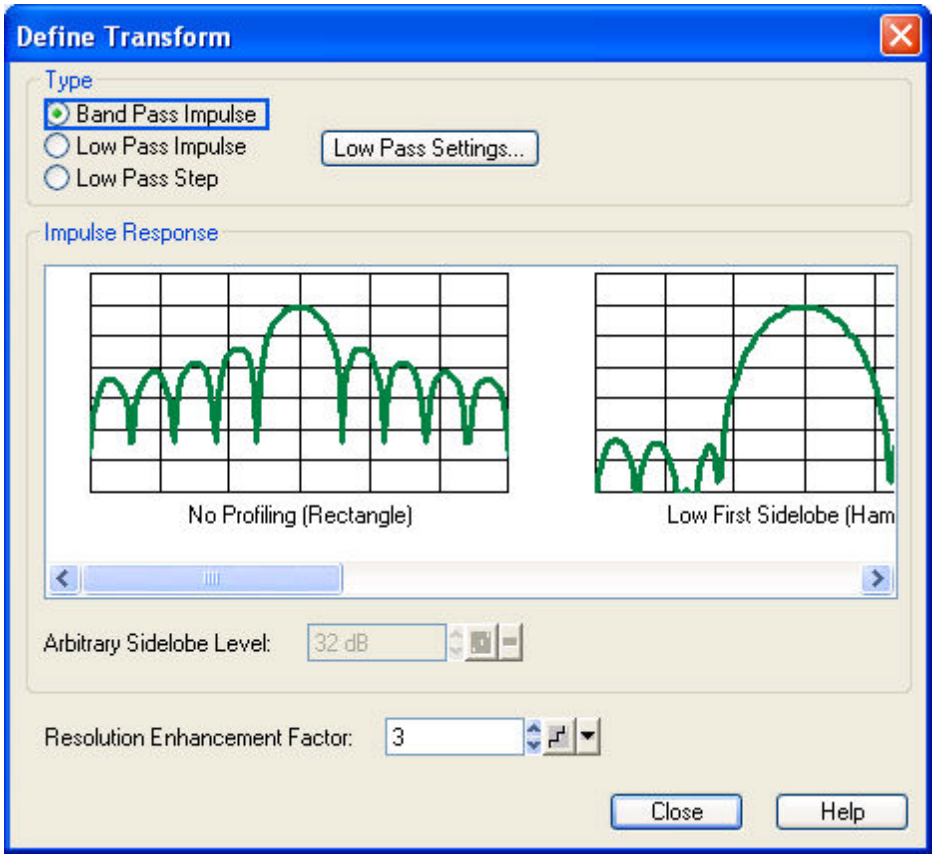
- For reflection measurements (S-parameters $S_{ii}$ or ratios with equal port indices) the distance between the source and the DUT is half the propagation time multiplied by the velocity of light in the vacuum times the velocity factor of the receiving port defined in the *Channel – Offset* menu *(Distance = 1/2 * Time * $c_0$ * Velocity Factor)*. The factor 1/2 accounts for the return trip from the DUT to the receiver.
- For transmission measurements, the distance is calculated as the propagation time times the velocity of light in the vacuum times the velocity factor of the receiving port defined in the *Channel – Offset* menu *(Distance = Time * $c_0$ * Velocity Factor)*.

Due to the properties of the Chirp z-transformation the trace is periodic in time and repeats after an unambiguous range of $\Delta t = 1/\Delta f$, where $\Delta f$ is the spacing between two consecutive frequency points. To extend the unambiguous range, either reduce the sweep span (*Channel – Stimulus*) or increase the number of sweep points.

**Remote control:**
```
CALCulate<Chn>:TRANsform:TIME:STARt
CALCulate<Chn>:TRANsform:TIME:STOP
CALCulate<Chn>:TRANsform:TIME:CENTer
CALCulate<Chn>:TRANsform:TIME:SPAN
CALCulate<Chn>:TRANsform:TIME:XAXis TIME | DISTance
```

## Define Transform

The *Define Transform* dialog selects the transformation type and the frequency domain window which is applied to the trace in order to optimize its time domain response.




- The radio buttons in the *Type* panel select a band pass or low pass transform. To calculate a low pass transform the sweep points must be on a harmonic grid (otherwise the analyzer will only be able to calculate an approximate result and generate a warning). *Low Pass Settings...* opens a dialog to establish or change a harmonic grid (not available for memory traces).
- The *Impulse Response* panel shows the impulse response of a constant trace over a finite sweep range (i.e. a rectangular function) that was filtered in the frequency domain using different windows. The selected window is applied to the active trace.
- If an *Arbitrary Sidelobes (Dolph-Chebychev)* window is selected, the *Arbitrary Sidelobe Level* (sidelobe suppression) can be set below the *Impulse Response* diagrams.
- The *Resolution Enhancement Factor* 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 resolution in time domain can be improved. The ideal resolution enhancement factor depends on the properties of the DUT. In distance to fault measurements on cables, factors between 3 and 5 turned out to be a good choice.

For a comparison of the different transformation types and windows and for application examples please also refer to the application note 1EZ44_OE which is posted on the R&S internet.

NOTE

*The frequency domain window is used to filter the trace prior to the time domain transformation. An independent Time Gate can be used after the transformation in order to eliminate unwanted responses.*

**Remote control:**
```
CALCulate<Chn>:TRANsform:TIME[:TYPE]
CALCulate<Chn>:TRANsform:TIME:STIMulus
CALCulate<Chn>:TRANsform:TIME:WINDow
CALCulate<Chn>:TRANsform:TIME:DCHebyshev
```

CALCulate<Chn>:TRANsform:TIME:RESolution:EFACtor

## Band Pass and Low Pass Mode

The analyzer provides two essentially 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 a generally 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.

The band pass and low pass modes are compared below.

| Transform type | Band pass | Low pass |
|---|---|---|
| Advantages | Easiest to use: works with any set of equidistant sweep points | Higher response resolution (doubled)<br>Includes information about DC value<br>Real result<br>Impulse and step response |
| Restrictions | No step response<br>Undetermined phase | Needs harmonic grid |
| Use for... | Scalar measurements where the phase is not needed<br>DUTs that don't operate down to f = 0 (e.g. pass band or high pass filters) | Scalar measurements where the sign is of interest<br>DUT's with known DC value |

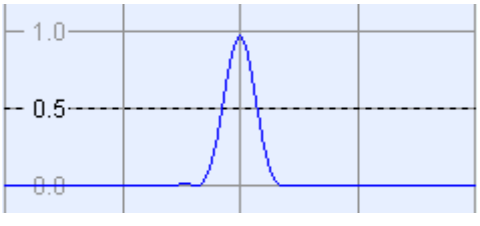
**Remote control:** CALCulate<Chn>:TRANsform:TIME[:TYPE]

## Impulse and Step Response

In low pass mode, the analyzer can calculate two different types of responses:

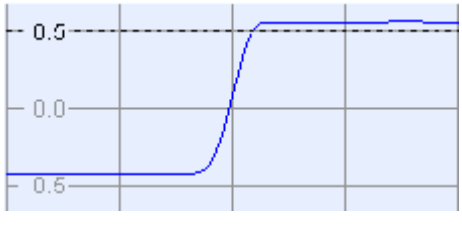
- The impulse response corresponds to the response of a DUT that is stimulated with a short pulse.
- The step response corresponds to the response of a DUT that is stimulated with a voltage waveform that transitions from zero to unity.

The two alternative responses are mathematically equivalent; the step response can be obtained by integrating the impulse response:




Integrate impulse response

Obtain step 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.

**Remote control:** `CALCulate<Chn>:TRANsform:TIME:STIMulus`

## 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 sidelobes (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:

| Window | Sidelobe suppression | Relative impulse width | Best for... |
|---|---|---|---|
| No Profiling (Rectangle) | 13 dB | 1 | – |
| Low First Sidelobe (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 sidelobe suppression |
| Steep Falloff (Bohman) | 46 dB | 1.9 | Dynamic range: separation of distant responses with different amplitude |
| Arbitrary Sidelobes (Dolph-Chebychev) | User defined between 10 dB and 120 dB | 1.2 (at 32 dB sidelobe suppression) | Adjustment to individual needs; tradeoff between sidelobe suppression and impulse width |

**Remote control:** `CALCulate<Chn>:TRANsform:TIME:WINDow`
`CALCulate<Chn>:TRANsform:TIME:DCHebyshev`

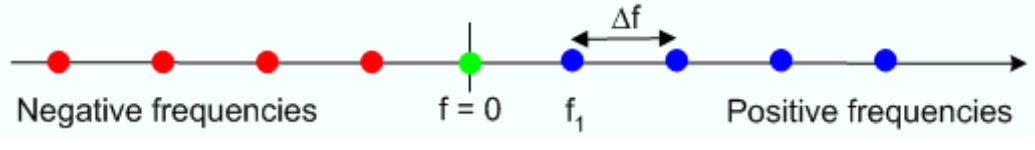
## Low Pass Settings

The *Low Pass Settings* dialog defines the harmonic grid for low pass time domain transforms.

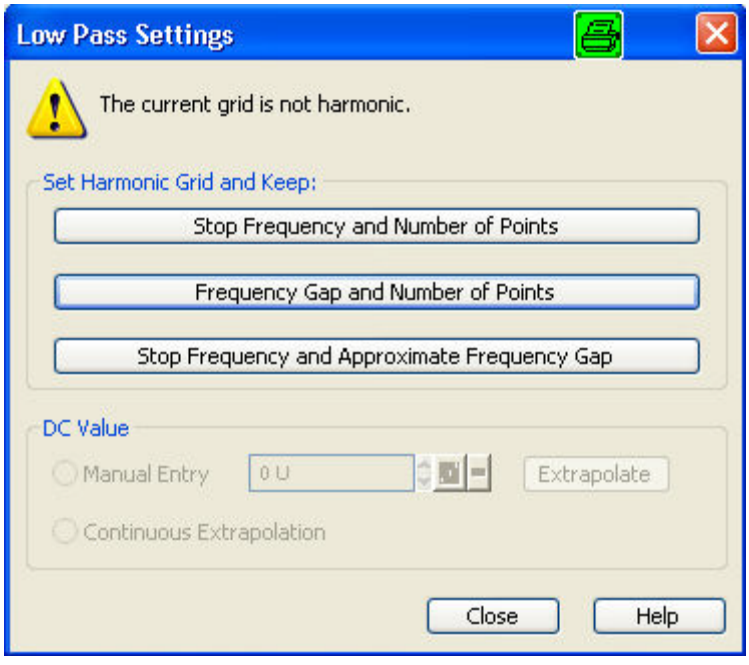
**Harmonic grid**

A harmonic grid is formed by a set of equidistant frequency points $f_i$ (i = 1...n) with spacing $\Delta f$ and the additional condition that $f_1 = \Delta f$. 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 with respect to the DC value makes harmonic grids suitable for lowpass time domain transformations.



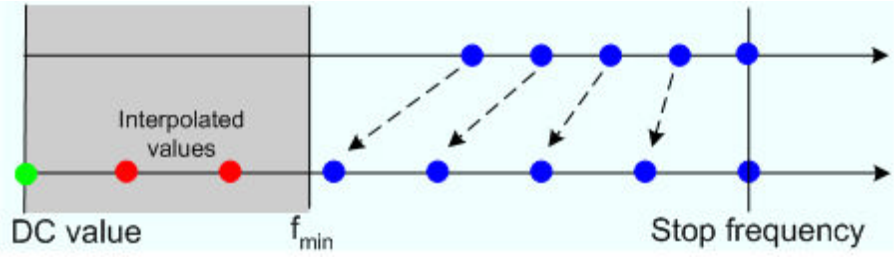

The dialog can be used to change the present grid of sweep points, which may or may not be harmonic.

- The three buttons in the *Set Harmonic Grid...* panel provide alternative algorithms for calculation of a harmonic grid, based on the current sweep points.
- The control elements in the *DC Value* panel 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 zero frequency and possibly at additional points in the range between f = 0 and f = $f_{min}$.

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 measured sweep points (blue dots) and the properties of the DUT.




- 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)* and let the analyzer calculate the remaining values (red dots) by linear interpolation of the magnitude and phase(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. If a cable with known termination is measured, then these real numbers should be entered as

DC values. ).Clicking the *Extrapolate* button will initiate an extrapolation of the measured trace towards f = 0 and overwrite the current DC value. This can be used for a consistency check.

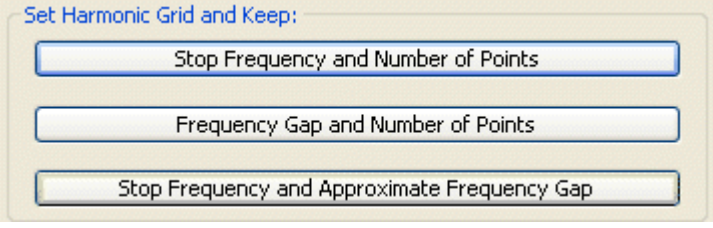
- *Continuous Extrapolation* initiates an extrapolation of the measured trace towards lower frequencies, so that the missing values (green and red dots) are obtained without any additional input. The extrapolation is repeated after each sweep.

**Remote control:**

```
CALCulate<Chn>:TRANsform:TIME:LPASs KFSTop | KDFRequency |
KSDFrequency
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
```

## Set Harmonic Grid

In the *Set Harmonic Grid...* panel of the *Low Pass Settings* dialog, a harmonic grid can be calculated in three alternative ways:



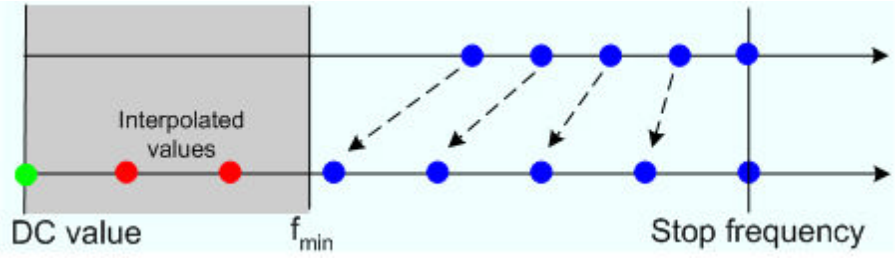

- Keep *Stop Frequency and Number of Points* calculates a harmonic grid based on the current stop frequency *(Channel – Stimulus – Stop)* and the current number of sweep points *(Channel – Sweep – Number of Points).* This may increase the step width, narrowing the unambiguous range but improving the time domain resolution.
- Keep *Frequency Gap and Number of Points* calculates a harmonic grid based on the current stop frequency *(Channel – Stimulus – Stop)* and the current frequency gap (the spacing between the equidistant sweep points, i.e. the sweep *Span* divided by the *Number of Points* minus one) . This leaves the unambiguous range unchanged but may lower the stop frequency and therefore deteriorate the time domain resolution.
- Keep *Stop Frequency and Approximate Frequency Gap* calculates a harmonic grid based on the current stop frequency *(Channel – Stimulus – Stop),* increasing the number of points *(Channel – Sweep – Number of Points)* in such a way that the frequency gap remains approximately the same. This leaves both the time domain resolution and the unambiguous range unchanged but may 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.

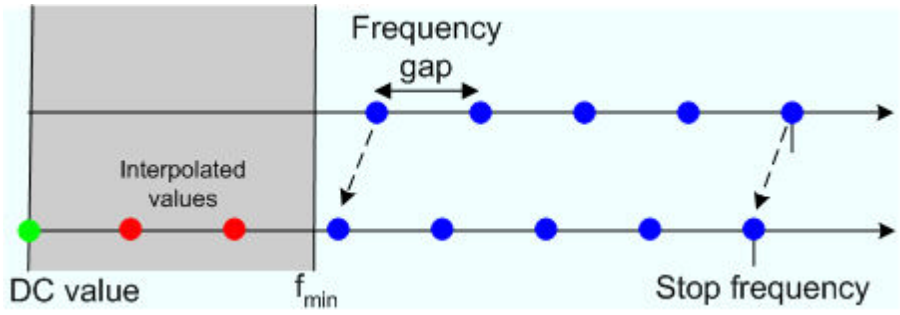
**Visualization of the harmonic grid algorithms**

The three types of harmonic grids have the following characteristics:

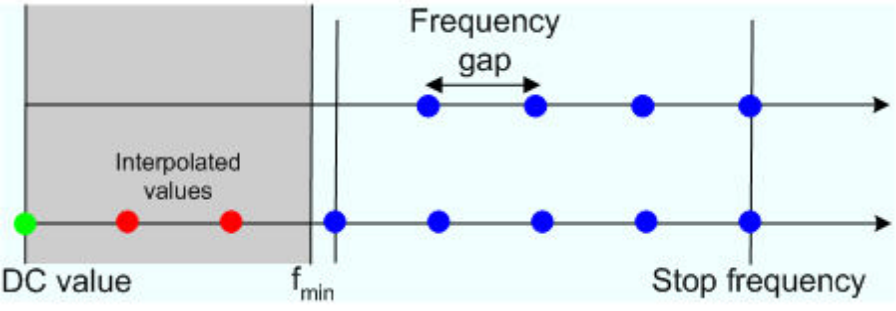
- Keep *Stop Frequency and Number of Points* means that the stop frequency and the number of sweep points is maintained. The sweep points are re-distributed across the range between the minimum frequency of the analyzer and the stop frequency; the step width may be increased.




- Keep *Frequency Gap and Number of Points* means that the number of sweep points and their relative spacing is maintained. The entire set of sweep points is shifted towards lower frequencies so that the stop frequency is decreased.




- 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.




NOTE *The figures above are schematic and do not comply with the conditions placed on the number of sweep points and interpolated/extrapolated values.*

NOTE *The harmonic grids can not 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 will 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.*

TIP **Finding the appropriate algorithm**

The three types of harmonic grids have different advantages and drawbacks. The following table helps you to find the appropriate grid.

| Grid type: Keep | Sweep time | Time domain resolution | Unambiguous range |
|---|---|---|---|
| Stop freq. and no. of points | → | ↑ | ↓ |
| Freq. gap and no. of points | → | ↓ | → |
| Stop freq. and approx. freq. gap | ↑ | → | → |

**Remote control:** `CALCulate<Chn>:TRANsform:TIME:LPASs KFSTop | KDFRequency | KSDFrequency`

## Time Gate

Switches the time gate defined via *Define Time Gate* on or off. An active time gate acts on the trace in time domain as well as in frequency domain representation. *Gat* is displayed in the trace list while the time gate is active.

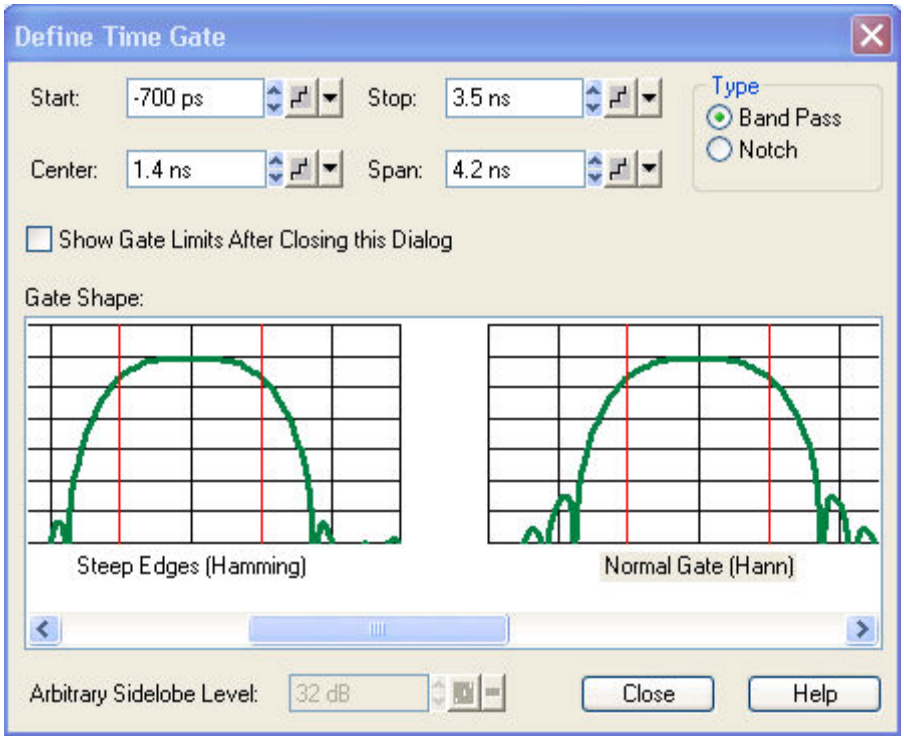
NOTE *The time gate is independent of the frequency window used to filter the trace prior to the time domain transformation.*

TIP In time domain representation, you can use the time gate settings in order 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.

**Remote control:** `CALCulate<Chn>:FILTer[:GATE]:TIME:STATe <Boolean>`

## Define Time Gate

The *Define Time Gate* defines the properties of the time gate used to eliminate unwanted responses that appear on the time domain transform.




- *Start* and *Stop* or *Center* and *Span* define the size of the time gate. The analyzer generates a warning if the selected time span exceeds the unambiguous range which is given by $\Delta t = 1/\Delta f$, where $\Delta f$ is the spacing between two consecutive frequency points. Simply reduce the time span until the warning disappears.
- The filter *Type* defines what occurs to the data in the specific time region. A *Band Pass* 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.
- If the *Show Gate Limits after Closing this Dialog* check box is selected two red lines indicating the start and stop of the time gate are permanently displayed in the diagram area.
- The *Gate Shape* panel 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.
- If an *Arbitrary Gate Shape (Dolph-Chebychev)* window is selected, the *Arbitrary Sidelobe Level* can be set below the *Gate Shape* diagrams.

Use the paste marker list for convenient entry of *Start* and *Stop* values.

**Comparison of time gates**

The properties of the time gates are analogous to the properties of the frequency domain windows. The following table gives an overview:

| Window | Sidelobe suppression | Passband ripple | Best for... |
|---|---|---|---|
| Steepest Edges (Rectangle) | 13 dB | 0.547 dB | Eliminate small distortions in the vicinity of the useful signal, if demands on amplitude accuracy are low |

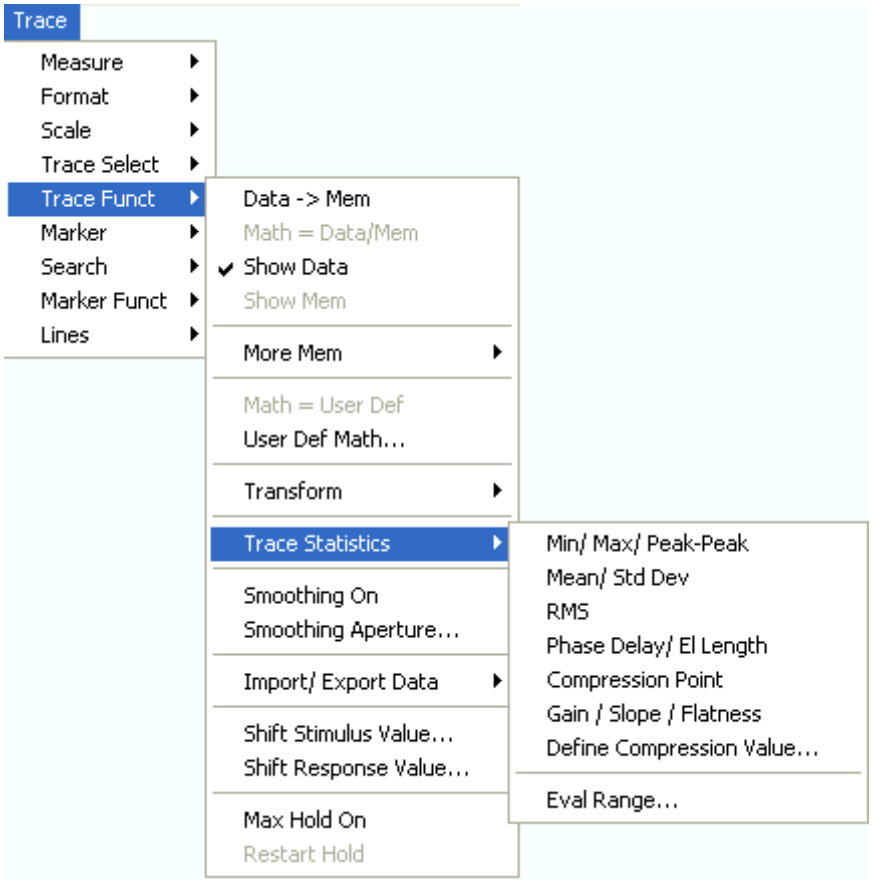
| | | | |
|---|---|---|---|
| Steep Edges (Hamming) | 43 dB | 0.019 dB | Good compromise between edge steepness and sidelobe suppression |
| Normal Gate (Hann) | 32 dB | 0.032 dB | Good compromise between edge steepness and sidelobe suppression |
| Maximum Flatness (Bohman) | 46 dB | 0 dB | Maximum attenuation of responses outside the gate span |
| Arbitrary Gate Shape (Dolph-Chebychev) | User defined between 10 dB and 120 dB | 0.071 dB | Adjustment to individual needs; tradeoff between sidelobe suppression and edge steepness |

**Remote control:**

```
CALCulate<Chn>:FILTer[:GATE]:TIME:STARt
CALCulate<Chn>:FILTer[:GATE]:TIME:STOP
CALCulate<Chn>:FILTer[:GATE]:TIME:CENTer
CALCulate<Chn>:FILTer[:GATE]:TIME:SPAN
CALCulate<Chn>:FILTer[:GATE]:TIME[:TYPE] BPASs |
NOTCh
CALCulate<Chn>:FILTer[:GATE]:TIME:SHOW
CALCulate<Chn>:FILTer[:GATE]:TIME:WINDow RECT |
HAMMing | HANN | BOHMan | DCHebyshev
CALCulate<Chn>:FILTer[:GATE]:TIME:DCHebyshev
```

## Trace Statistics

Opens a submenu to evaluate and display statistical and phase information of the entire trace or of a specific evaluation range and calculate the x-dB compression point.




- *Min/Max/Peak-Peak* displays or hides the essential statistical parameters of the trace in the selected evaluation range.
- *Mean/Std Dev* displays or hides the arithmetic mean value and the standard deviation of the trace in the selected evaluation range.
- *RMS* displays or hides the RMS value of the trace in the selected evaluation range.
- *Phase Delay/El Length* displays or hides the phase delay and the electrical length of the trace in the selected evaluation range *(Eval Range...).*
- *Compression Point* starts the x-dB compression point evaluation
- *Define Compression Value...* sets the compression level (x dB).
- *Gain/Slope/Flatness* displays or hides trace parameters that the analyzer calculates by means of two markers.
- *Eval Range...* opens a dialog to define the range for the statistical and phase evaluation and for the compression point measurement.

## Statistical Evaluation

The first three commands in the *Trace Statistics* submenu display or hide the maximum *(Max.)*, minimum *(Min.),* the peak-to-peak value *(Pk-Pk),* arithmetic mean value *(Mean)*, the standard deviation *(Std. Dev.)*, and the *RMS* value of all response values of the trace in the selected evaluation range *(Eval Range...)*.

**Definition of statistical quantities**

The statistical quantities are calculated from all response values in the selected 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).

- *Mean* is the arithmetic mean value of all response values:

$$Mean = \frac{1}{n}\sum_{i=1}^{n} y_i$$

- *Std. Deviation* is the standard deviation of all response values:

$$Std.Dev. = \sqrt{\frac{1}{n}\sum_{i=1}^{n}(y_i - \frac{1}{n}\sum_{i=1}^{n} y_i)^2}$$

- *Max.* and *Min.* are the largest and the smallest of all response values $y_i$.
- RMS is the root mean square (effective value) of all response values:

$$RMS = \sqrt{\frac{1}{n}\sum_{i=1}^{n} y_i^2}$$

- *P-P* is the peak-to-peak value and is equal to the difference *Max. – Min.*

NOTE

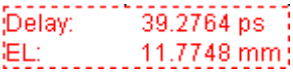
*The definitions given above apply to all linear, scalar trace formats. For a dB Mag trace, the Mean, Std. Dev. and RMS are calculated before logarithmation.*

**Remote control:**

```
CALCulate<Chn>:STATistics[:STATe]
CALCulate<Chn>:STATistics:RESult? MIN | MAX | PEAK2P | MEAN |
STDDev | RMS
CALCulate<Chn>:STATistics:MMPTpeak[:STATe]
CALCulate<Chn>:STATistics:MSTDev[:STATe]
CALCulate<Chn>:STATistics:RMS[:STATe]
```

## El. Length/Phase Delay

Displays or hides the phase delay *(Delay)* and the electrical length *(EL)* of the trace in the selected evaluation range *(Eval Range...)*. The parameters are only available for trace formats that contain phase information, i.e. for the formats *Phase, Unwrapped Phase*, and the polar diagram formats *Polar, Smith, Inverted Smith*. Moreover, the sweep type must be a *frequency sweep.*




**Definition of phase parameters**

The phase parameters are obtained from an approximation to the derivative of the phase with respect to frequency in the selected evaluation range.

- *Delay* 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 $\Delta f$ is the width of the evaluation range and $\Delta\Phi$ is the corresponding phase change. See also note on transmission and reflection parameters below.

- *EL* is the electrical length, which is 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 see mathematical relations.

If a dispersive connector type (i.e. a waveguide; see *Offset Model* dialog) is assigned to a test port related to a particular quantity, then the dispersion effects of the connector are taken into account for the calculation of the phase delay and the electrical length.

*To account for the propagation in both directions, the delay and the electrical length of a reflection parameter is only half the delay and the electrical length of a transmission parameter. The formula for PD above is for transmission parameters. See also introduction to section Channel – Offset.*

*The phase parameters are available only if the evaluation range contains at least 3 measurement points.*

The phase evaluation can cause misleading results if the evaluation range contains a ±360 deg phase jump. The trace format *Unwrapped Phase* avoids this behavior.

**Remote control:**
```
CALCulate<Chn>:STATistics[:STATe]
CALCulate<Chn>:STATistics:RESult? ELENgth | PDELay
CALCulate<Chn>:STATistics:EPDelay[:STATe]
```

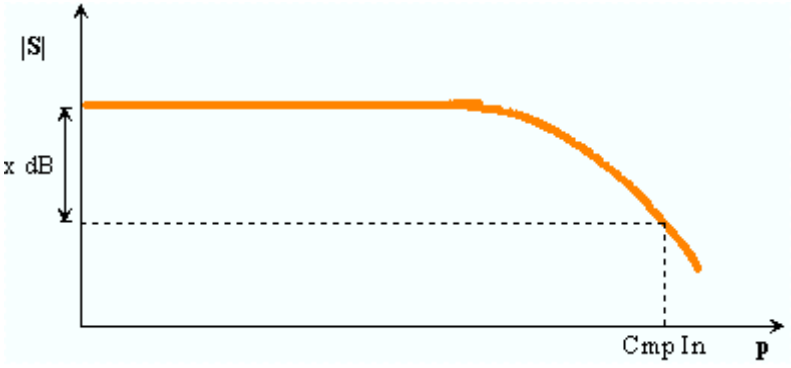
## Compression Point

Displays or hides all results related to the x-dB compression point of the trace, where x is the selected compression value. To obtain a valid compression point results, a power sweep must be active, the trace format must be *dB Mag* and either an S-parameter or a ratio must be measured.

**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). As an approximation for the small-signal value, the analyzer uses the value at the start level of the evaluation range *(Eval Range...)*.




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 ($|a_n|$ —> $x^*|a_n|$ ⇨ $|b_n|$ —> $x^*|b_n|$, so that the magnitude of all S-parameters remains constant).
When *Compression Point* is activated, a marker labeled *Cmp* is placed to compression point with the smallest stimulus level. Moreover the *Trace Statistics* info field shows the numerical results of the compression point measurement:

- *Cmp In* is the stimulus level at the compression point in units of dBm.
- *Cmp Out* is the magnitude of the wave reflected or transmitted by the DUT. If an S-parameter $S_{ij}$ is measured, then *Cmp Out* is the magnitude of the wave $b_i$.
- The ratio *Cmp Out* / *Cmp In* (in the example above: –14 dB) is equal to the magnitude of the measured S-parameter or ratio at the compression point.

The info field shows invalid results ('---') if the wrong sweep type, trace format or measured quantity is selected, or if the trace contains no x-dB compression points in the selected evaluation range.

**Remote control:** `CALCulate:STATistics:NLINear:COMP[:STATe]`
`CALCulate:STATistics:NLINear:COMP:RESult?`

## Define Compression Value

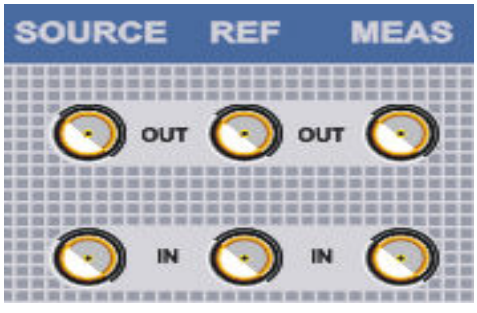
Opens the numeric entry bar to define the compression value x (in dB) for the compression point measurement.

**Remote control:** `CALCulate:STATistics:NLINear:COMP:LEVel`

## Gain/Slope/Flatness

Displays or hides trace parameters that the analyzer calculates by means of two markers placed on the active trace.

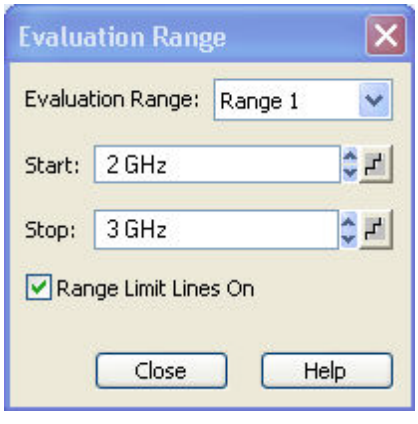
- *Gain* is the larger of the two stimulus values of marker 1 and 2.
- *Slope* is the difference of the stimulus values of marker 2 minus marker 1.
- *Flatness* is a measure of the deviation of the trace between the markers from linearity. The analyzer calculates the difference trace between the active trace and the straight line between marker 1 and marker 2. The flatness is the difference between the largest and the smallest response value of this difference trace.




**Remote control:** `CALCulate<Chn>:STATistics:SFLatness[:STATe]`
`CALCulate<Chn>:STATistics:RESult? GAIN | SLOPe | FLATness`

## Eval Range

Opens a dialog to define the range for the statistical and phase evaluation and for the x-dB compression point measurement. The evaluation range is a continuous interval of the sweep variable.




It is possible to select, define and display up to ten different evaluation ranges for each setup. *Full Span* means that the search range is equal to the *sweep range.* The statistical and phase evaluation and the compression point measurement take into account 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 identical to the marker search ranges. For more information see Search Range Dialog.*

**Remote control:** `CALCulate<Chn>:STATistics:DOMain:USER <numeric_value>`
`CALCulate<Chn>:STATistics:DOMain:USER:STARt <numeric_value>`
`CALCulate<Chn>:STATistics:DOMain:USER:STOP <numeric_value>`

## Smoothing On

Activates the smoothing function for the active trace, which may be a data or a memory trace. With active smoothing function, each measurement point is replaced by the arithmetic mean value of all measurement points located in a symmetric interval centered on the stimulus value. The width of the smoothing interval is referred to as the *Smoothing Aperture* and can be adjusted according to the properties of the trace.

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.

**Remote control:** `CALCulate<Chn>:SMOothing[:STATe] <Boolean>`

## Smoothing Aperture

Defines how many measurement points are averaged to smooth the trace if smoothing is switched on. The *Smoothing Aperture* is entered as a percentage of the total sweep span.

An 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. The average is calculated for every measurement point. Smoothing does not significantly increase the measurement time.

**Finding the appropriate aperture**

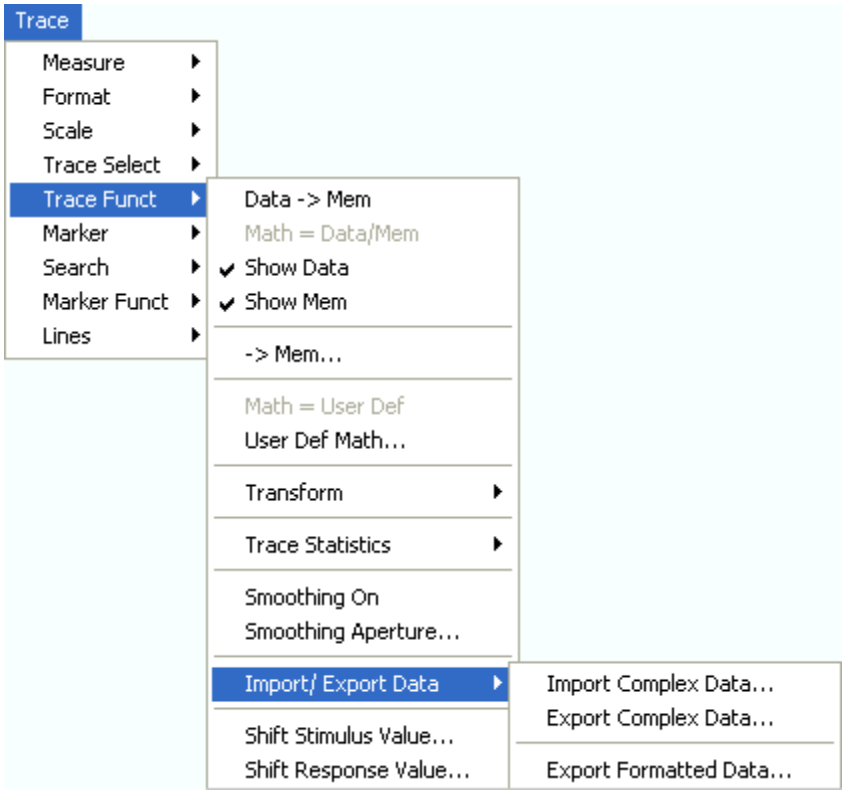
A large smoothing aperture enhances the smoothing effect but may also average out quick variations of the measured values and thus produce misleading results. To avoid errors, observe the following recommendations.

- Start with a small aperture and increase it only as long as you are certain that the trace is still correctly reproduced.
- As a general rule, the smoothing aperture should be 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 control:** `CALCulate<Chn>:SMOothing:APERture <numeric_value>`

## Import/Export Data

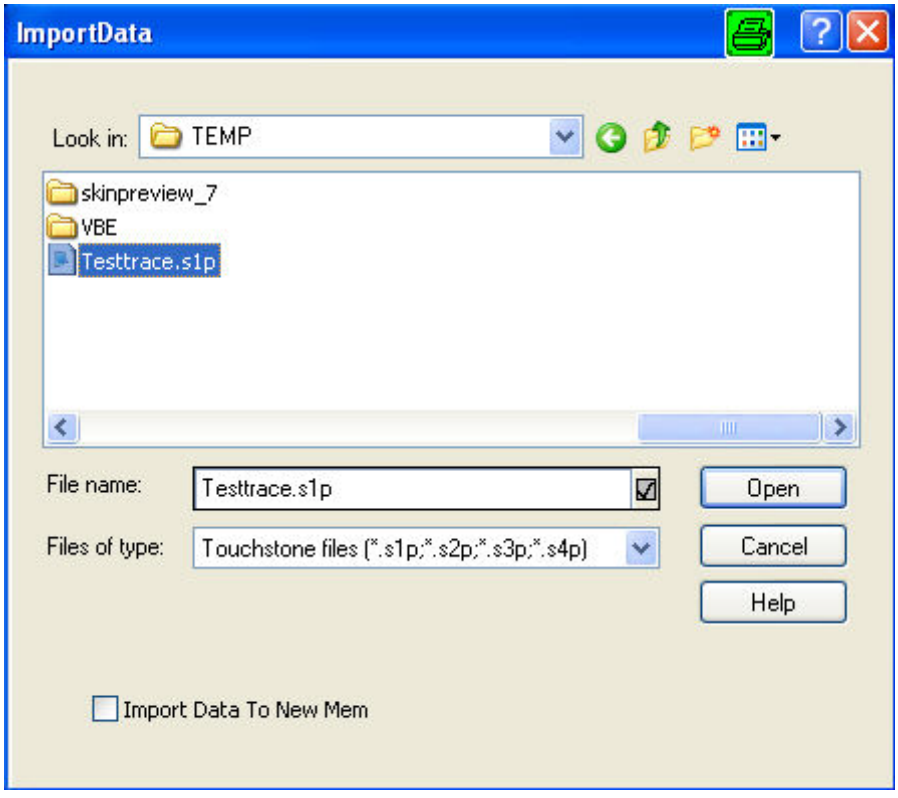
Stores one or several data or memory traces to a file or loads a memory trace from a file. Import or export is selected from a submenu:




- *Import Complex Data...* calls up an *Open File* dialog to load a memory trace from a trace file.
- *Export Complex Data...* calls up a *Save As...* dialog to store data or memory traces to a trace file.
- *Export Formatted Data...* calls up a *Save As...* dialog to store data or memory traces with their current trace format and stimulus values to a trace file.

### Import Complex Data...

Calls up a dialog to load a memory trace from a trace file. Trace files are ASCII files with selectable file format.

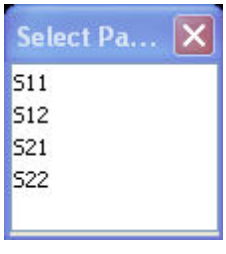



The loaded trace data is used to generate a memory trace which is coupled to the active data trace. *Import Data* corresponds to a standard *Open File* dialog with an additional checkbox:

- *Import Data to New Mem* qualifies whether the loaded data overwrite the active memory trace (box unchecked, analogous to *Data -> Mem*) or whether they are used to generate a new memory trace (box checked, analogous to *Data -> New Mem*).

**Trace file import and import conditions**

On loading data from a trace file, the analyzer displays a list box to select one of the traces stored in the file. The example below shows a *Touchstone* (*.s2p) file, containing all four 2-port S-parameters; see *Trace File Formats.*




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. Compatibility 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 files contains no compatible data.

When a *.s1p file is opened the *Select Parameter* box indicates the 1-port (reflection) parameter

S11, irrespective of the actual S-parameter stored in the file.

To import a trace file (*.snp or *.csv) you can also use the Windows Explorer and simply double-click the file or drag and drop the file into the NWA application. The imported data generates a memory trace which is coupled to the active data trace.

**Remote control:** `MMEMory:LOAD:TRACe "<trc_name>","<file_name>"`

## Export Complex Data...

Calls up a dialog to store data or memory traces to a trace file. Trace files are ASCII files with selectable file format.

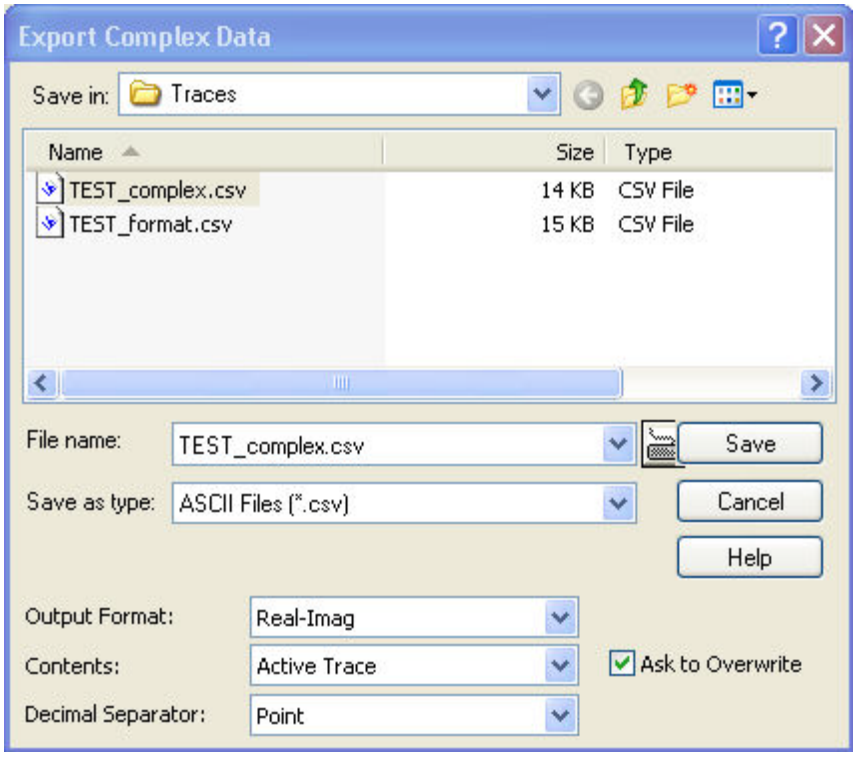
**Complex vs. Formatted Data**

Complex and formatted data is tapped at different stages of the data processing chain.

- *Complex* trace values are the raw values at the beginning of the trace data flow. None of the following stages (trace mathematics, shift response, time domain gate, trace formatting and smoothing) affects the complex data. *Export Complex Data* writes the raw stimulus values (frequency/power/time, according to the sweep type) and the raw, complex measurement points to a file.
- *Formatted* trace values are the fully processed trace values displayed on the analyzer screen. All possible stages of the trace data flow (trace mathematics, shift response, time domain gate, trace formatting and smoothing) are taken into account. *Export Formatted Data* writes the displayed stimulus values (e.g. a time scale if a time domain transformation is used) and the displayed measurement points to a file.

Data export can serve many purposes, e.g.:

- To process and evaluate measurement data in an external application.
- To store measurement data and re-import it in a future measurement session.




*Export Complex Data* corresponds to a standard *Save As...* dialog with an additional panel to specify the export *Options*. The export options are remembered when the dialog is closed. The *Dec. Separator* export option is not available for Matlab (*.dat) file export. Touchstone files *.s<n>p contain either a single trace (*.s1p) or the complete set of S-parameters of an n-port and use the point as *Dec. Separator;* see *Trace File Formats*.

- Output Format selects the format for the exported trace data. In the *Real - Imag* format, each complex measurement value is represented by its real and imaginary part. It is also possible to store the linear magnitude and phase *(lin Mag-Phase)* or dB magnitude and phase *(dB Mag-Phase)* of each value.
- Content selects only the *Active Trace* or *All Traces of (the) Active Channel* (including all data and memory traces) for data export to an ASCII (*csv) file.
- Dec. Separator selects either the *Point* or the *Comma* (if needed to process the exported data with an external application) as a separator for decimal numbers exported to an ASCII (*.csv) file.
- Ask to Overwrite activates a message box to be displayed before an older trace file with the same file name and directory is overwritten.

**Selecting an appropriate file format**

Use a **Touchstone** file format to export a fixed number of S-parameter data traces to a file that can be evaluated with applications such as Agilent's Microwave Design System (MDS) and Advanced Design System (ADS). The data must be acquired in a frequency sweep.

- *.s1p files are intended for a 1-port S-parameter (the reflection coefficient S11) but can be used as well to store any single S-parameter. Note that any S-parameter will be labeled S11 in the file and in the *Select Parameter* box.
- *.s<n>p files (<n> = 2, 3,, 4) 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.

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.
- Use export options.

Use a the **Matlab (*.dat)** format if you want to import and process the trace data in Matlab.

For more information see Trace File Formats.

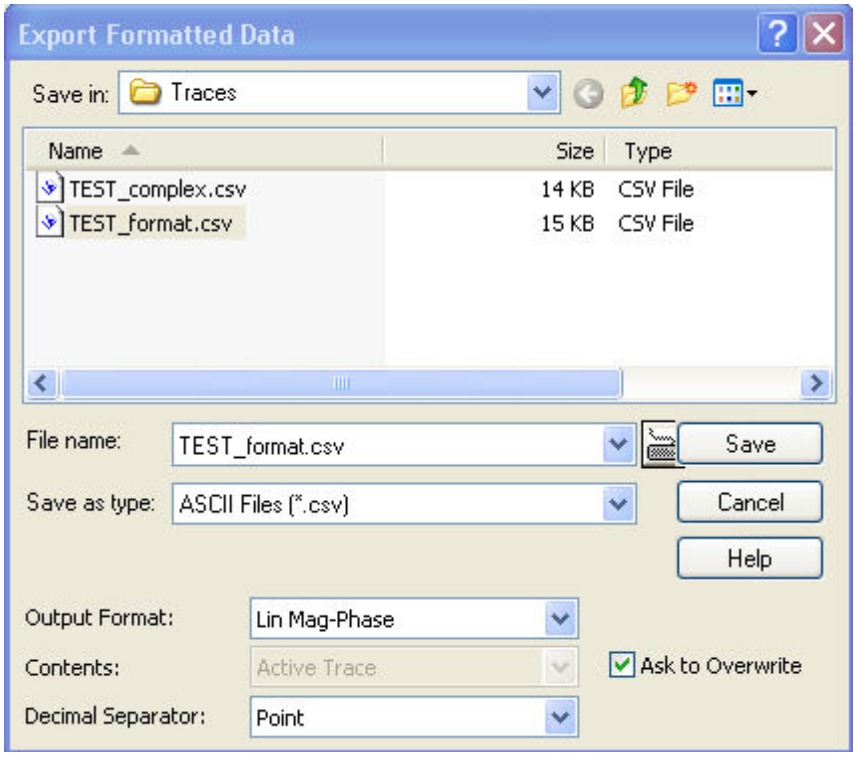
**Remote control:** `MMEMory:STORe:TRACe "<trc_name>","<file_name>"[,UNFormatted]`

## Export Formatted Data...

Calls up a dialog to store data or memory traces with their current trace format and stimulus values to a trace file. Trace files are ASCII files with selectable file format. Only the displayed values of the trace (e.g. the logarithmic magnitude, if trace format *dB Mag* is selected) are exported. Therefore the trace file doesn't necessarily contain the full (complex) information about the trace.

*The exported formatted values correspond to the results in the marker info field. 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. See also Complex vs. Formatted Data.*

An exported formatted trace can be processed and evaluated in an external application. Re-import into the analyzer is only possible if the complex trace values are available.




*Export Formatted Data* corresponds to a standard *Save As...* dialog with an additional panel to specify the export *Options.* The *Dec. Separator* export option is not available for Matlab (*.dat) file export.

*Touchstone files *.s<n>p contain either a single trace (*.s1p) or the complete set of S-parameters*

*of an n-port and use the point as Dec. Separator; see Trace File Formats; they are only available if complex trace values are exported (trace formats Polar, Smith, Inverted Smith). Moreover, touchstone files cannot be used to export mathematical traces.*

The following output options also depend on the trace format.

- Output Format selects the format for the exported trace data. In the *Re/Im* format, each complex measurement value is represented by its real and imaginary part. It is also possible to store the linear magnitude and phase *(lin Mag-Phase)* or dB magnitude and phase *(dB Mag-Phase)* of each value.
- Content selects only the *Active Trace* or *All Traces of (the) Active Channel* (including all data and memory traces) for data export to an ASCII (*csv) file.
- Dec. Separator selects either the *Point* or the *Comma* (if needed to process the exported data with an external application) as a separator for decimal numbers exported to an ASCII (*.csv) file.
- Ask to Overwrite activates a message box to be displayed before an older trace file with the same file name and directory is overwritten.

**Selecting an appropriate file format**

Use a **Touchstone** file format to export a fixed number of S-parameter data traces to a file that can be evaluated with applications such as Agilent's Microwave Design System (MDS) and Advanced Design System (ADS). The data must be acquired in a frequency sweep.

- *.s1p files are intended for a 1-port S-parameter (the reflection coefficient S11) but can be used as well to store any single S-parameter. Note that any S-parameter will be labeled S11 in the file and in the *Select Parameter* box.
- *.s<n>p files (<n> = 2, 3,, 4) 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.

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.
- Use export options.

Use a the **Matlab (*.dat)** format if you want to import and process the trace data in Matlab.

For more information see Trace File Formats.

**Remote control:** `MMEMory:STORe:TRACe "<trc_name>","<file_name>",FORMatted`

## Trace File Formats

Trace files are ASCII files with selectable file format. The analyzer provides two classes 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 *Selecting an appropriate file format.*

## Touchstone files

All Touchstone files contain a header, a comment section, and the actual trace data:

```
# HZ  S  RI  R   50.0000
! Rohde & Schwarz ZVA
! Measurement: S11
! 2003-07-07
!
60297750.000000        0.498113  -0.054290
80297000.000000        0.504888  -0.081229
...
```

The header consists of the following data elements:

# specifies beginning of header line (required at top of file). <Frequency unit> HZ / KHZ / MHZ / GHZ allowed for imported files. The analyzer always uses HZ for exported data.

<Data file type> at present: S for S-parameter files.

<Data format> RI for Re/Im, MA for lin. Mag-Phase, DB for dB Mag-Phase. The data format for export files can be selected in the Export Data dialog.

<Normalizing impedance> Impedance system in which the data was defined. The analyzer uses 50.0000 Ω.

Comment lines start with the exclamation mark (!) and may contain any text used for documentation of the trace data file. Any number of comment lines may be inserted before or after the header line.

The trace data depend on the number of ports <n> and the data format. For real and imaginary values (data format = RI) the trace data for each stimulus frequency are arranged as follows:

- **1-port files (*.s1p)**

Freq Re(S11) Im(S11)

S11 can be replaced by an any S-parameter, so the *s1p format is suitable for exporting an arbitrary data trace representing an S-parameter.

- **2-port files (*.s2p)**

Freq Re(S11) Im(S11) Re(S21) Im(S21) Re(S12) Im(S12) Re(S22) Im(S22)

(all values arranged in 1 line)

- **3-port files (*.s3p)**

Freq Re(S11) Im(S11) Re(S12) Im(S12) Re(S13) Im(S13)
Re(S21) Im(S21) Re(S22) Im(S22) Re(S23) Im(S23)
Re(S31) Im(S31) Re(S32) Im(S32) Re(S33) Im(S33)

(values arranged in 3 lines)

- **4-port files (*.s4p)**

Freq Re(S11) Im(S11) Re(S12) Im(S12) Re(S13) Im(S13) Re(S14) Im(S14)
Re(S21) Im(S21) Re(S22) Im(S22) Re(S23) Im(S23) Re(S24) Im(S24)
Re(S31) Im(S31) Re(S32) Im(S32) Re(S33) Im(S33) Re(S34) Im(S34)
Re(S41) Im(S41) Re(S42) Im(S42) Re(S43) Im(S43) Re(S44) Im(S44)

(values arranged in 4 lines)

The stimulus frequencies are arranged in ascending order. If a *lin. Mag-Phase (MA)* or *dB Mag-Phase (DB)* data format is selected the real and imaginary S-parameter values $Re(S_{ij})$, $Im(S_{ij})$ are replaced by lin $Mag(S_{ij})$, $phase(S_{ij})$ or dB $Mag(S_{ij})$, $phase(S_{ij})$, respectively.

## 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.

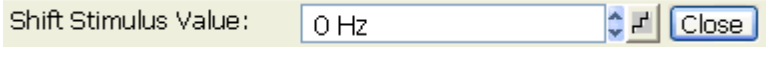
<imTrace2>... second response value of second trace: im<Trace_Name> for output format Re/Im, ang<Trace_Name> for output formats lin. Mag-Phase or dB Mag-Phase. The data format for export files can be selected in the Export Data dialog. first response value of second trace. HZ / KHZ / MHZ / GHZ allowed for imported files. The analyzer alway 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. A semicolon is also inserted before the end of each line.

The stimulus values are arranged in ascending order.

## Shift Stimulus Value...

Shifts the active trace in horizontal direction, leaving the positions of all markers unchanged. The positive or negative offset value for the stimulus variable is entered into an input field. The unit depends on the sweep type.




*Shift Stimulus Value* can be used in Cartesian as well as in polar diagrams. The visible effect depends on the diagram type:

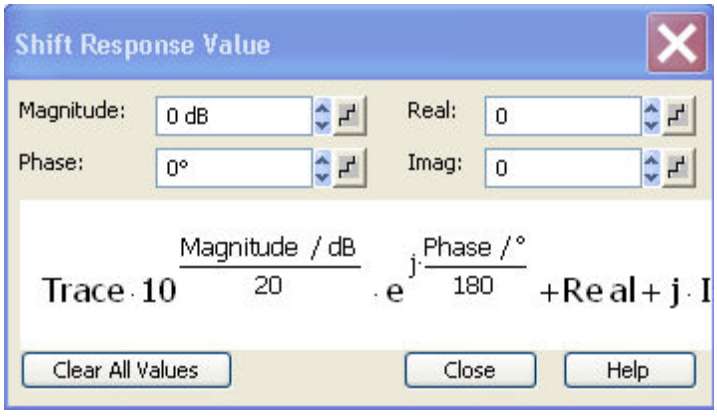
- In Cartesian diagrams, the trace is shifted relative to the markers and the x-axis.
- In polar diagrams, the trace is not affected, however, markers change their position.

Use a negative offset value to reset a shifted trace to its original position.

**Remote control:** `DISPlay:WINDow<Wnd>:TRACe<WndTr>:X:OFFset <numeric_value>`

## Shift Response Value...

Modifies all points of the active trace by means of an added and/or a multiplied complex constant. The constant complex values are entered in the four input fields of the *Shift Response Value* dialog.




The units of the constants are adjusted to the format of the active trace. Setting all values to zero *(Clear All Values)* restores the original trace.

**Effect of the constants**

The trace points are modified according to the formula displayed in the lower part of the dialog:

$$M_{new} = M_{old} \cdot 10^{<\text{Magnitude}>/20\,\text{dB}} \cdot e^{j\cdot<\text{Phase}>/180^\circ} + <\text{Real}> + j\,<\text{Imag}>$$

The formula and the different constants are adjusted to the different display formats of a trace:

- The *Magnitude* 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* added constant shifts a real trace in vertical direction, leaving the imaginary part unchanged.
- The *Imaginary* added constant shifts a imaginary trace in vertical direction, leaving the real part unchanged.

Shifting the trace by means of constant values is a simple case of trace mathematics. Use the *Define Math* dialog to define more complicated mathematical operations.

**Remote control:** `DISPlay:WINDow<Wnd>:TRACe<WndTr>:Y:OFFset <Magnitude>[, <Phase>, <Real>, <Imag>]`

**Remote control:** `CALCulate<Chn>:MATH:WUNit:STATe ON | OFF`

## Max Hold On

Enables (if selected) or disables the max hold (peak hold) function for the active trace. With enabled max hold function, the displayed trace shows the maximum values that the analyzer acquired since the start of the measurement. The max hold trace is real, it is based on the magnitude of the trace values

(the phase values are discarded).

The max hold process can be restarted any time using *Restart Hold.* It is also restarted automatically when the channel or trace settings are changed so that the previous measurement results are no longer valid.

**Remote control:** `CALCulate<Chn>:PHOLd MAX | OFF`

### Restart Hold

Restarts the max hold (peak hold) function for the active trace, discarding the old maximum measurement results. This softkey has no effect unless *Max Hold On* is active.

**Remote control:** `CALCulate<Chn>:PHOLd MAX | OFF`

## Marker

The functions in the *Marker* menu are used to position markers on a trace, configure their properties and select the format of the numerical readout.

**Markers**

Markers are tools 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, which may 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, reference marker, delta marker or discrete marker.

**Marker types**

- A (normal) marker*(Mkr 1, Mkr 2, ...)* determines the coordinates of a measurement point on the trace. Up to 10 different normal markers can be assigned to a trace.
- A reference marker*(Ref)* defines the reference value for all delta markers.
- A delta marker*(Δ)* indicates the coordinates relative to the reference marker.
- The stimulus value of a *discrete marker* always coincides with a sweep point.

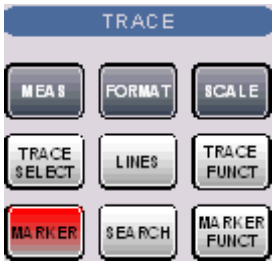
A special set of markers Mk1 to Mkr4 is provided for bandfilter search mode.

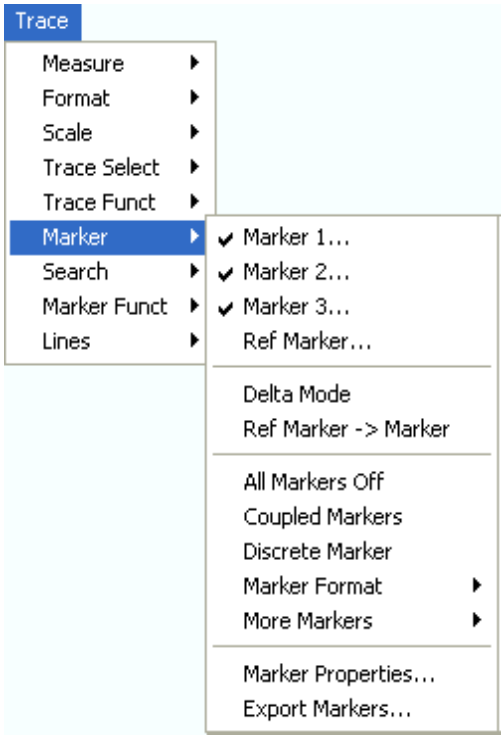
The most common tasks to be performed with markers can be achieved using the *Marker* menu functions:

- 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 Funct.*).
- Search for specific points on the trace (*Search*).




The *Marker* menu contains the following functions:

- *Marker 1/2/3* creates the markers numbered 1, 2, and 3.
- *Ref. Marker...* creates the reference marker which is used to measure relative values and distances.
- *Delta Mode* activates the display of the active marker values relative to the reference marker.
- *Ref. Marker -> Marker* places the reference marker to the position of the active marker.
- *All Markers Off* removes all markers from all traces of the active setup.
- *Coupled Markers* couples the markers on different traces.
- *Discrete Marker* turns the active marker into a discrete marker and vice versa.
- *More Markers* opens a submenu to create the markers numbered 4 to 10.
- *Marker Format* defines an output format for the (complex) marker values.
- *More Markers* opens a submenu to create the markers numbered 4 to 10.
- *Marker Properties...* opens a dialog with extended marker settings.
- *Export Markers...* exports the marker values to an ASCII file.

**Toggle functions in the *Marker* menu**

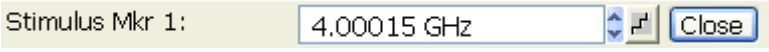
Some of the *Marker* functions toggle between two alternative states when they are pressed repeatedly:

- *Delta Mode* switches the delta mode for the active marker on or off.
- *Coupled Mkrs* activates or deactivates marker coupling.
- *Marker 1, ..., Marker 10* and *Ref. Marker* create a marker or remove it from the display. A removed marker remembers its properties (stimulus value, format, delta mode, number) and will be restored with these properties when *Marker <n>* or *Ref. Marker* is selected again. The marker properties are definitely lost if the associated trace is deleted.

Markers are available for all diagram types (Trace – Format).

## Marker 1/2/3

Creates the markers numbered 1, 2, and 3, respectively, and assigns them to the active trace (toggle function). *Marker 1/2/3* opens the numeric entry bar to define the marker position *(Stimulus Mkr 1/2/3)*. The default position is the center of the sweep range.




On closing the *Stimulus Mkr 1/2/3* numeric entry bar a marker symbol (triangle) labeled *Mkr <n>* is positioned on the trace and the marker coordinates are displayed in the *Info Field*.

**Activating and moving markers**

To select one of several markers as an active marker, do one of the following:

- Click the marker symbol.
- Click the marker line in the marker info field.

To change the position of the active marker on the trace use one of the following methods:

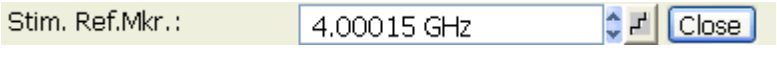
- Drag-and-drop the marker symbol to the desired position.
- Click the *Marker <n>* or *Ref. Marker* softkey to call up the entry bar for the new stimulus value.
- Right-click the diagram area or select *Mkr. Properties* to call up the *Marker Properties* dialog and select the new stimulus value.
- Use the *Search* functions to place the marker to a specific point on the trace.

*If the marker position is defined explicitly by entering a numeric value, the marker position can be outside the sweep range. If it is just varied using the rollkey, the mouse or the cursor keys, it always remains within the sweep range. If the position of a marker outside the sweep range is varied, it is automatically moved to the start or stop value of the sweep range, whichever is closer.*

**Remote control:**
```
CALCulate<Chn>:MARKer<Mk>[:STATe] ON
CALCulate<Chn>:MARKer<Mk>:Y?
```

## Ref. Marker

Creates a reference marker and assigns it to the active trace (toggle function). *Ref. Marker* opens the numeric entry bar to define the marker position *(Stimulus Ref Mkr)*. The default position is the center of the sweep range.




On closing the *Stimulus Ref Mkr* numeric entry bar a marker symbol (triangle) labeled *Ref* is positioned on the trace and a line indicating *Ref* plus the marker coordinates is inserted in the marker *info field*.

The reference marker defines the reference value for all markers that are in *Delta Mode*.

**Remote control:**
```
CALCulate<Chn>:MARKer<Mk>:REFerence[:STATe] ON
CALCulate<Chn>:MARKer<Mk>:REFerence:Y?
```

## Delta Mode

Converts the active marker to a delta marker so that its values are measured and indicated relative to the reference marker (toggle function). A Δ sign placed in front of the marker line indicates that the marker is in *Delta Mode.*

The reference marker itself can not be set to delta mode but must be present when another marker is set to delta mode. The analyzer takes into account these conditions when *Delta Mode* is selected:

- If *Delta Mode* is selected while the reference marker is active, the marker in the info list after the reference marker is activated and set to delta mode. If the current trace contains the reference marker only, a new *Mkr 1* is created and set to delta mode.
- If *Delta Mode* is selected for a normal marker while the current trace contains no reference marker, a reference marker is created.

*A delta marker is required to set the sweep span using marker functions (Span = Marker).*

**Remote control:** `CALCulate<Chn>:MARKer<Mk>:DELTa[:STATe] ON`

## Ref. Marker -> Marker

Places the reference marker to the position of the active marker. As a consequence, the active marker takes the role of a reference marker.

*Ref. Marker -> Marker* is not active if the active marker is a reference marker.

**Remote control:** –

## All Markers Off

Removes all markers from all traces of the active setup. The removed markers remember their properties (stimulus value, format, delta mode, number) when they are restored. The marker properties are definitely lost if the associated trace is deleted.

**Remote control:** `CALCulate<Chn>:MARKer<Mk>:AOFF`

## Coupled Markers

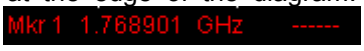
Couples the markers of all traces in the active setup to the markers of the active trace (toggle function). While marker coupling is active, the active trace markers assume the role of master markers; the other markers behave as slave markers, following any change of position of the master marker.

**Effects of marker coupling**

The concept of marker coupling means that corresponding markers on different traces (i.e. markers with the same number or reference markers) are positioned to the same stimulus values but keep their independent format and type settings. When a trace with markers is selected as the active trace and marker coupling is switched on, the following happens:

- The active trace and all associated markers are left unchanged. The active trace markers become the master markers of the setup.
- Markers on the other traces which have no corresponding master marker are removed but

remember their properties and can be re-activated after the coupling is released.

- The remaining markers on the other traces become slave markers and are moved to the position of the corresponding master markers. "Missing" slave markers are created so that each trace has the same number of markers placed at the same position.
- If the position of a master marker is outside the sweep range of the slave trace, the slave marker is displayed at the edge of the diagram. The marker info field indicates an invalid measurement result: 


While marker coupling is active, it is possible to:

- Move a master marker and thus change the position of all corresponding slave markers.
- Activate another trace in order to make the associated markers the new master markers.

NOTE Marker coupling makes sense only if the master and the slave traces use the same stimulus variable. Channels with a different stimulus variable (sweep type) are not coupled.

**Remote control:** `CALCulate<Chn>:MARKer<Mk>:COUPled[:STATe] ON`

## Discrete Marker

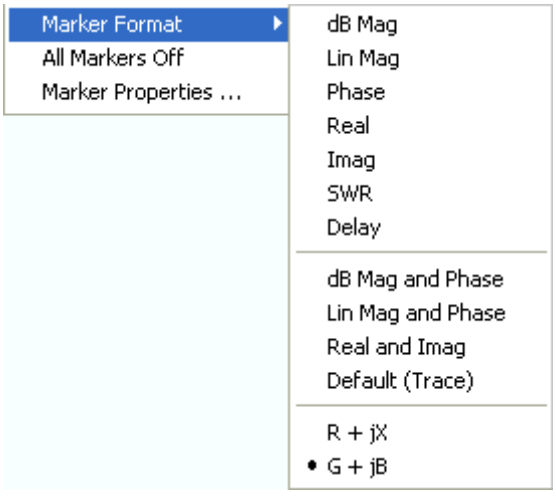
Turns the active marker into a discrete marker and vice versa.

TIP The stimulus value of a discrete marker always coincides with a sweep point. Use discrete markers in order to avoid that the marker indicates an interpolated measurement value.

**Remote control:** `CALCulate<Chn>:MARKer<Mk>:MODE`

## Marker Format

Opens a submenu to select an output format for the (complex) active marker value in the marker info field. The default marker format is the format of the associated trace. The current format is indicated with a ●.




All marker formats are available irrespective of the measured quantity. The output values are calculated

by a simple conversion of a complex measurement result, where the marker format defines the conversion rules. This flexibility in the calculation must be kept in mind when interpreting the results and physical units displayed; see also Measured Quantities and Display Formats.

**Short description of marker formats**

The formats of the markers assigned to a trace are independent of each other and of the trace format settings. The following table gives an overview on how a complex marker value z = x + jy is converted.

| Marker Format | Description | Formula |
|---|---|---|
| dB Mag | Magnitude of z in dB | \|z\| = sqrt ( $x^2 + y^2$ )<br>dB Mag(z) = 20 * log\|z\|<br>dB |
| Lin Mag | Magnitude of z, unconverted | \|z\| = sqrt ( $x^2 + y^2$ ) |
| Phase | Phase of z | $\phi$ (z) = arctan (y/x) |
| 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\|) |
| Delay | Group delay, neg. derivative of the phase response*) | – d $\phi$ (z) / d $\omega$ |
| dB Mag and Phase | Magnitude of z in dB and phase in two lines | 20 * log\|z\| dB<br>arctan ( Im(z) / Re(z) ) |
| Lin Mag and Phase | Magnitude of z (unconverted) and phase in two lines | \|z\|<br>arctan ( Im(z) / Re(z) ) |
| Real and Imag | Real and imaginary part of z in two lines | x<br>y |
| Default (Trace) | Marker format identical with trace format | – |
| R + j X | Unnormalized resistance and reactance; L or C in three lines<br>(Smith diagram) | R<br>X<br>L or C**) |
| G + j B | Unnormalized conductance and susceptance; L or C in three lines<br>(Inverted Smith diagram) | G<br>B<br>L or C**) |

*) The delay aperture is defined in the Trace – Format menu.

**) The equivalent inductances or capacitances L or C are calculated from the imaginary part of the impedance according to

$$L = \frac{1}{\omega} X \quad \text{if } X \geq 0$$

$$C = \frac{1}{\omega \cdot X} \quad \text{if } X < 0$$

**Remote control:** `CALCulate<Chn>:MARKer<Mk>:FORMat ...`

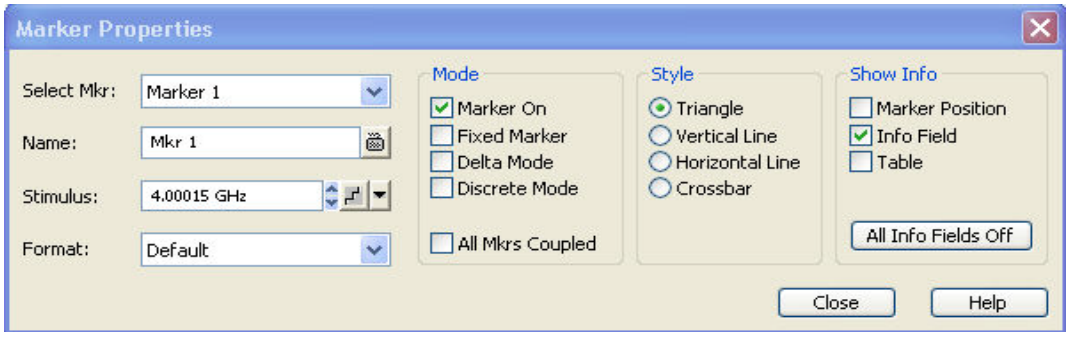
## More Markers

Opens a submenu to create the markers numbered 4 to 10. The markers are analogous to marker no. 1 to 3.

**Remote control:** `CALCulate<Chn>:MARKer<Mk>[:STATe] ON`

## Marker Properties...

Opens a dialog to define the properties of all markers of the active trace.




In the left part the dialog contains four input fields and drop-down lists to select a marker and define the basic properties of a selected marker which is switched on in the *Mode* panel.

- *Select Mkr.* opens a drop-down list to select the *Ref. Marker* or one of the *Marker 1, ... 10* that may be associated to the active trace.
- *Name* defines a marker name which may exceed the length of the input box and contain letters, numbers, blanks and special characters.
- *Stimulus* defines the stimulus value (in Cartesian diagrams: the x-axis position) of the marker.
- *Format* defines the output format of the marker value.

The *Mode* panel contains check boxes to select properties that are related to the marker positions. All properties can be combined.

- *Marker On* displays the selected marker in the diagram and in the marker info field or removes it. Marker configurations are only available while the marker is switched *On*. A removed marker remembers its properties (stimulus value, format, delta mode, number ...) when it is switched on again. The marker properties are definitely lost if the associated trace is deleted.
- *Fixed Marker* freezes thecurrent response value of the selected marker. The marker can still be shifted horizontally but the vertical position remains fixed if the other marker settings are changed. Markers must be inside the sweep range and have a valid response value when they are fixed.
- *Delta Mode* sets the selected marker to delta mode and displays its values relative to the reference marker.
- *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.
- *All Mkrs Coupled* couples the markers of all traces in the active setup to the markers of the active trace; see *Coupled Markers*.

*Style* qualifies how the selected marker is displayed on the screen.

*Show Info* selects the marker info to be displayed at the *Marker Position,* in the marker *info field* or in a separate *Table* below the diagram area. The display options may be selected simultaneously or all switched off. The table provides more information than the marker info field:

| Marker | Trace | Stimulus | Response | Delta | Discr | Fixed | Tracking | Search Range |
|---|---|---|---|---|---|---|---|---|
| Ref | Trc1 | 3.440171000 GHz | -6.426 dB | ☐ | ☑ | ☑ | Off | Full Range |
| Mkr 1 | Trc1 | 4.000150000 GHz | -5.364 dB | ☐ | ☐ | ☐ | Off | Full Range |

**Remote control:**
```
CALCulate<Chn>:MARKer<Mk>[:STATe] ON
CALCulate<Chn>:MARKer<Mk>:TYPE NORMal | FIXed
CALCulate<Chn>:MARKer<Mk>:DELTa[:STATe] ON
CALCulate<Chn>:MARKer<Mk>:MODE CONTinuous | DISCrete
CALCulate<Chn>:MARKer<Mk>:COUPled[:STATe] ON
```

### 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 setup 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 setup with two traces *Trc1* and its memory trace. *Trc1* has no markers assigned, the memory trace has four markers named *Mkr 1, ..., Mkr 4.*

```
Trc1 S21
No Markers

Mem2[Trc1] S21
Mkr 1 969.60000 MHz  -4.896 dB
Mkr 2 129.72115 MHz  -7.896 dB
Mkr 3 4.793734 GHz  -7.896 dB
Mkr 4 2.461727 GHz  -5.192 dB
```

**Remote control:**
```
MMEMory:STORe:MARKer "file_name"
```

## Search

The *Search* menu uses markers to locate specific points on the trace.

**Search functions**

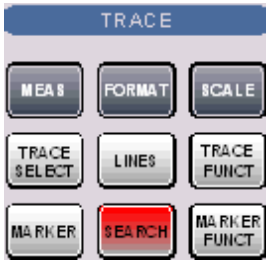
The search functions are tools for searching measurement data according to specific criteria. A search consists of analyzing the measurement points of the current trace (or of a user-defined subrange termed the S*earch Range*) in order to find one of the following:

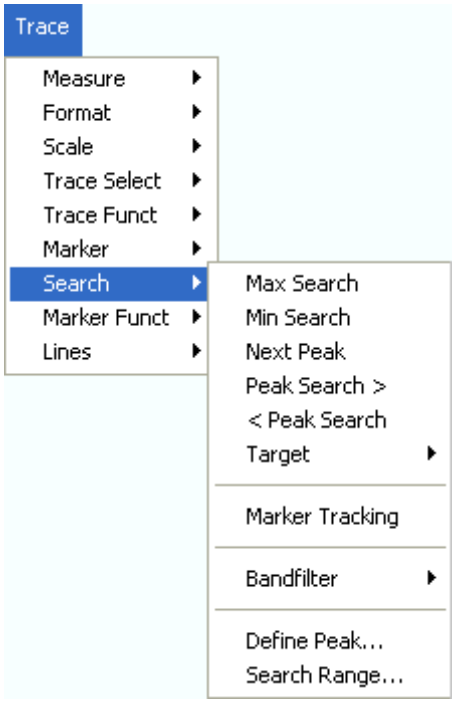
- Absolute or relative (local) maxima and minima (minimum/maximum search).
- Trace points with a specific response value (target search).
- Trace segments with a shape that is characteristic for bandpass or bandstop filters (bandfilter search).

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 *Mkr 1* 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 in order to find all possible search results. Moreover the analyzer provides a *Tracking* mode where the search is repeated after each sweep.




- *Max Search* sets the active marker to the absolute maximum in the search range.
- *Min Search* sets the active marker to the absolute minimum in the search range.
- *Next Peak* sets the active marker to the next maximum or minimum in the search range, depending on the current search criterion.
- *Peak Search >* sets the active marker to the next peak with higher stimulus value.
- *< Peak Search* sets the active marker to the next peak with lower stimulus value.
- *Target* opens a submenu to search for a specific value on the trace.
- *Marker Tracking* causes the search to be repeated after each sweep.
- *Bandfilter* opens a submenu to search for trace segments with characteristic bandfilter shape and calculate the filter parameters.
- *Define Peak* defines the criteria for the peak search.
- *Search Range...* assigns a search range to each marker of the current trace.

The search functions are available in all Cartesian and polar diagram types (see Trace – Format). In polar diagrams *(Polar, Smith, Inverted Smith),* where complex values are displayed, the magnitude of the response values provides the search criterion.

## Max Search

Sets the active marker to the absolute maximum in the search range, i.e. to the largest 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 magnitude.

By default the search range coincides with the sweep range. If the active trace contains no markers, a

marker *Mkr 1* is created to indicate the search result.

**Remote control:**
```
CALCulate<Chn>:MARKer<Mk>:FUNCtion:EXECute MAXimum
CALCulate<Chn>:MARKer<Mk>:FUNCtion:RESult?
```

## Min Search

Sets the active marker to the absolute minimum in the search range, i.e. to the 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 minimum magnitude.

By default the search range coincides with the sweep range. If the active trace contains no markers, a marker *Mkr 1* is created to indicate the search result.

**Remote control:**
```
CALCulate<Chn>:MARKer<Mk>:FUNCtion:EXECute MINimum
CALCulate<Chn>:MARKer<Mk>:FUNCtion:RESult?
```

## Next Peak

Sets the active marker to the next maximum or minimum in the search range, depending on the current search criterion.

- If a *Max Search* or a *Bandpass Search* 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 a *Min Search* or a *Bandstop Search* 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.

By default the search range coincides with the sweep range. If the active trace contains no markers, a marker *Mkr 1* is created to indicate the search result. *Next Peak* is disabled while a *Target Search* is active.

**Remote control:**
```
CALCulate<Chn>:MARKer<Mk>:FUNCtion:EXECute NPEak
CALCulate<Chn>:MARKer<Mk>:FUNCtion:RESult?
```

## Peak Search >

Sets the active marker to the next peak with higher stimulus value. If the active trace contains no markers, a marker *Mkr 1* created to indicate the search result. The peak criteria are defined via Define Peak.

**Remote control:**
```
CALCulate<Chn>:MARKer<Mk>:FUNCtion:EXECute RPEak
CALCulate<Chn>:MARKer<Mk>:FUNCtion:RESult?
```

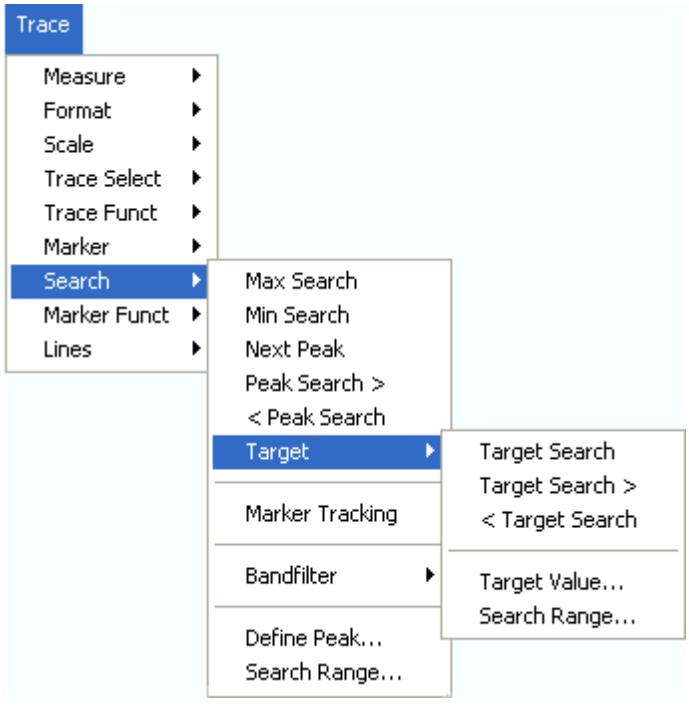
## < Peak Search

Sets the active marker to the next peak with lower stimulus value. If the active trace contains no markers, a marker *Mkr 1* created to indicate the search result. The peak criteria are defined via Define Peak.

**Remote control:** `CALCulate<Chn>:MARKer<Mk>:FUNCtion:EXECute LPEak`
`CALCulate<Chn>:MARKer<Mk>:FUNCtion:RESult?`

## Target

Opens a submenu to search for a specific value on the trace.




- *Target Search* activates the search for the target value.
- *Target Search >* activates the search for the target value to the right of the active marker.
- *< Target Search* activates the search for the target value to the left of the active marker.
- *Target Value* specifies the target value.
- *Search Range* confines the search to a subrange of the sweep.

### Target Search

Activates the search and sets the active marker to the defined target value. If the active trace contains no markers, a marker *Mkr 1* is created to indicate the search result.

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 *Target Search >* function.

If the target is not found (e.g. because the active trace doesn't contain the target value), then the active marker is not moved away from its original position.

**Remote control:** `CALCulate<Chn>:MARKer<Mk>:FUNCtion:EXECute TARGet`
`CALCulate<Chn>:MARKer<Mk>:FUNCtion:RESult?`

### Target Search >

Activates the search to the right of the active marker position and sets the active marker to the defined target value. The target search range is between the active marker position and the end (Stop) of the sweep range. If the active trace contains no markers, a marker *Mkr 1* is created to indicate the search result, and the target search range starts at the beginning (Start) of the sweep range.

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 *Target Search* > repeatedly.

If the target is not found (e.g. because the active trace doesn't contain the target value), then the active marker is not moved away from its original position.

**Remote control:**
```
CALCulate<Chn>:MARKer<Mk>:FUNCtion:EXECute RTARget
CALCulate<Chn>:MARKer<Mk>:FUNCtion:RESult?
```

### < Target Search

Activates the search to the left of the active marker position and sets the active marker to the defined target value. The target search range is between the (Start) of the sweep range and the active marker position. If the active trace contains no markers, a marker *Mkr 1* is created to indicate the search result, and the target search range starts at the end (Stop) of the sweep range.

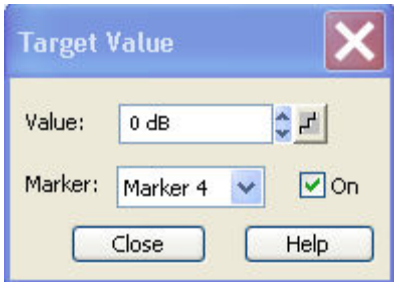
If the target value occurs at several stimulus values, the marker is placed to the search result with the smallest stimulus value. The other measurement points with the same target value can be located using *Target Search* > repeatedly.

If the target is not found (e.g. because the active trace doesn't contain the target value), then the active marker is not moved away from its original position.

**Remote control:**
```
CALCulate<Chn>:MARKer<Mk>:FUNCtion:EXECute LTARget
CALCulate<Chn>:MARKer<Mk>:FUNCtion:RESult?
```

### Target Value

Opens an dialog to select a marker for the target search and define the target value.




It is possible to define up to ten different target values for each trace and assign them to the markers no. 1 to 10. The input fields in the *Target Search* dialog are used to select the markers and define the associated search ranges:

- *Marker* selects one of the ten markers that can be assigned to the trace. If a selected marker does not exist, it is created as soon as *On* is checked. A created marker is displayed in the center of the search range.
- *Value* selects the target value to be assigned to the selected marker. The target *Value* is entered with the

unit of the active trace.

**Remote control:** `CALCulate<Chn>:MARKer<Mk>:TARget`

### Search Range...

Opens the *Search Range Dialog* to confine the target search to a subrange of the sweep.

## Define Peak

Opens a dialog to define the peak type to be searched for.

It is possible to select the peak type up to ten times for each trace and assign the selection to the markers no. 1 to 10.

- *Marker* Selects one of the ten markers that can be assigned to the trace. If a selected marker does not exist, it is created as soon as *On* is checked. A created marker is displayed in the center of the search range.

The radio buttons in the *Peak Type* panel offer the following alternative peak types:

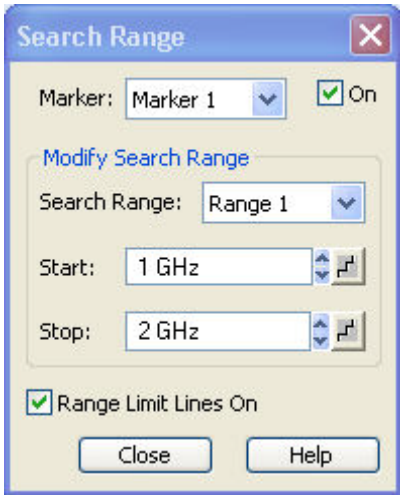
- Local Max activates the peak search for local maxima only. The response value at a local maximum is larger than the values in the vicinity.
- Local Min activates the peak search for local minima only. The response value at a local minimum is smaller than the values in the vicinity.
- Local Min or Max activates the peak search for both local maxima and minima.

## Search Range

Opens the *Search Range Dialog* to confine the minimum/maximum search to a subrange of the sweep.

### Search Range Dialog

Defines search ranges for the maximum/minimum or target search.




It is possible to define up to ten different search ranges for each setup and assign them to the markers no. 1 to 10. The input fields in the *Search Range* dialog are used to select the markers and define the associated search ranges:

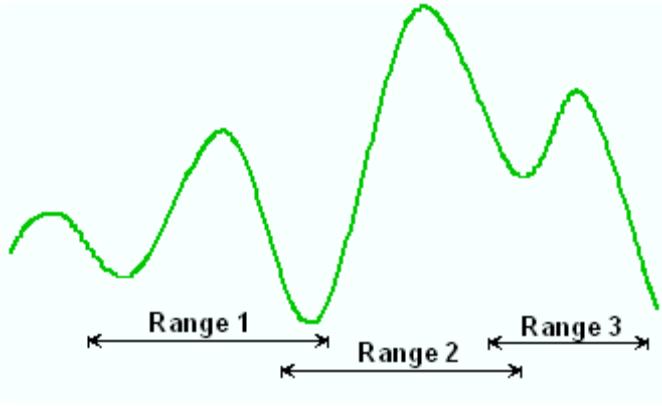
- *Marker* Selects one of the ten markers that can be assigned to a trace in the current setup. If a selected marker does not exist, it is created as soon as *On* is checked. A created marker is displayed in the center of the search range.
- *Search Range* Selects the search range to be assigned to the 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.
- *Start* defines the beginning of the search range. *Start* must be smaller than the *Stop* value, otherwise the search will not be initiated.
- *Stop* defines the end of the search range. *Stop* must be larger than the *Start* value, otherwise the search will not be initiated.
- *Range Limit Lines On* displays two vertical lines indicating the *Start* and the *Stop* value of the current search range in the diagram area. This function is enabled as soon as one of the search ranges 1 to 10 is selected.

**Search range properties**

In contrast to the marker properties defined in the *Marker* and *Search* menus, the ten search ranges are valid for the entire setup. This means that, once defined, each of them can be assigned to any marker in the setup, irrespective of the trace and channel that the marker belongs to.

The default search range of each new marker is *Full Span.* The analyzer provides the greatest flexibility in defining search ranges. In particular, two search ranges may 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 ranges can be used to search a trace for several local maxima.




Use the paste marker list for convenient entry of *Start* and *Stop* values.

**Remote control:** `CALCulate<Chn>:MARKer<Mk>:FUNCtion:DOMain:USER`
`CALCulate<Chn>:MARKer<Mk>:FUNCtion:DOMain:USER:STARt`
`CALCulate<Chn>:MARKer<Mk>:FUNCtion:DOMain:USER:STOP`

## Marker Tracking

Causes the active minimum/maximum or target search of the active marker to be repeated after each sweep: When tracking mode is active the marker typically changes its horizontal and its vertical position as the measurement goes on.

**Tracking mode properties**

The tracking mode is available for all search modes, i.e. for minimum/maximum search, target search and bandfilter search. The *Marker Tracking* function in the *Search* submenu is valid for an active minimum/maximum and target search; bandfilter tracking can be activated separately. Tracking is marker-specific but can be switched on for several markers simultaneously.

Switching between *Max Search* and *Min Search* does not affect the tracking mode.Tracking is switched off though when one of the following actions is performed:

- Redefinition of the active marker position by means of the functions in the *Marker* menu or drag-and-drop of the active marker symbol.
- Change of the active marker's search mode, e.g. from a minimum/maximum search to a target search.

*Tracking* is a toggle function: Selecting the function repeatedly switches the tracking mode on and off.

**Remote control:** `CALCulate<Chn>:MARKer<Mk>:SEARch:TRACking`

## Bandfilter

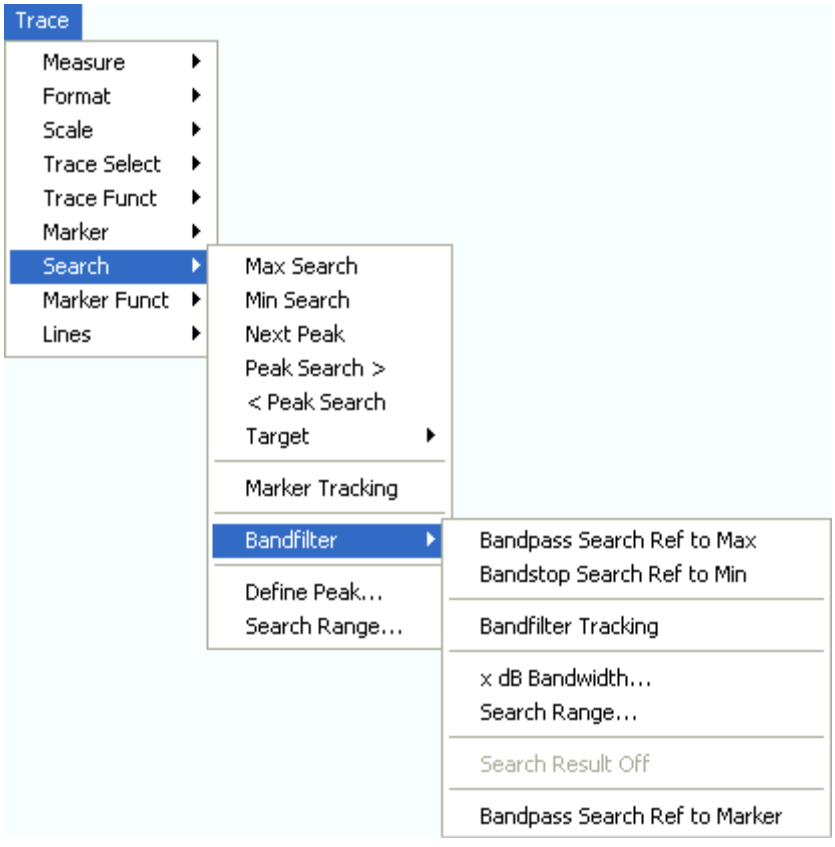
Opens a submenu to search for trace segments with a bandpass or bandstop shape and determine characteristic filter parameters.

**Bandfilter search and 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 *x dB Bandwidth*.
- A bandstop region contains a local minimum around which the magnitude of the trace increases by more than a specified *x dB Bandwidth*.

The analyzer locates bandpass and bandstop regions and determines their position *(Center* frequency) and shape *(Bandwidth, LBE, UBE*, Quality factor *Q*; see *Show Results).* For a meaningful definition of the *x dB Bandwidth* criterion, the trace format must be *dB Mag.*




- *Bandpass Search Ref to Max* activates the search for a bandpass region in the active trace, starting at the absolute maximum of the active trace within the search range.
- *Bandstop Search* activates the search for a bandstop region in the active trace, starting at the absolute minimum of the active trace within the search range.
- *Bandfilter Tracking* causes the bandfilter search to be repeated after each sweep.
- *x dB Bandwidth...* sets the level defining the filter bandwidth.
- *Search Range...* confines the search to a subrange of the sweep.
- *Search Result Off* hides the display of the bandfilter parameters in the diagram area.
- Bandpass Search Ref to Marker activates the search for a bandpass region in the active trace, starting at the position of the active marker.

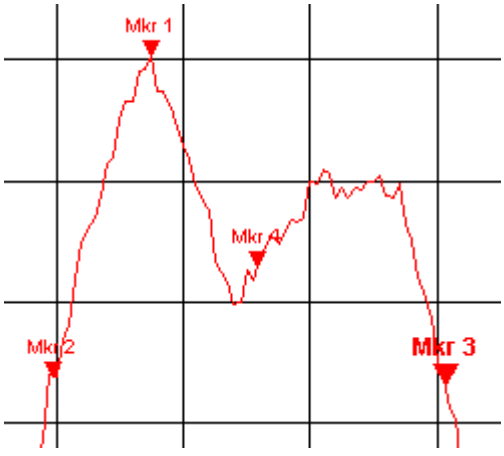
NOTE

*Bandfilter mode can be selected for a broad range of measured quantities (Trace – Measure), provided that the display format is dB Mag. To obtain real filter parameters, the measured quantity must be a transmission S-parameter and a frequency sweep must be performed. For other quantities (e.g reflection parameters), the Bandfilter functions are still useful to analyze general trace properties. In some display formats (e.g. Phase) the bandfilter search is disabled.*

## Bandpass Search Ref to Max

Activates the search for a bandpass region on the active trace and activates bandfilter *Tracking*. A bandpass region is the tallest peak in the search range with a minimum excursion specified by means of the *x dB Bandwidth* parameter.

When *Bandpass Search* is activated the analyzer uses (or creates) the four markers *Mkr 1* to *Mkr 4* to locate the **bandpass region:**




- *Mkr 1* indicates the maximum of the peak.
- *Mkr 2* indicates the point on the left edge of the peak where the trace value is equal to the maximum minus *x dB Bandwidth.*
- *Mkr 3* indicates the point on the right edge of the peak where the trace value is equal to the maximum minus *x dB Bandwidth.*
- *Mkr 4* indicates the center of the peak, calculated as the arithmetic mean value of the *LBE* and *UBE* positions.

For a symmetric peak, the positions of *Max* and *Center* coincide.

The band filter search results are displayed in the bandfilter info field.

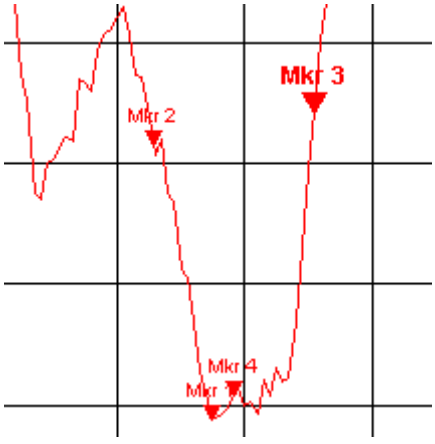
To search for a bandpass region in the vicinity of the active marker, use Bandpass Search Ref to Marker.

**Remote control:** `CALCulate<Chn>:MARKer<Mk>:FUNCtion:BWIDth:MODE BPASs`
`CALCulate<Chn>:MARKer<Mk>:FUNCtion:EXECute BFILter`

## Bandstop Search

Activates the search for a bandstop region on the active trace and activates bandfilter *Tracking*. A bandstop region is the lowest peak (local minimum) in the search range with a minimum excursion that is specified by means of the *x dB Bandwidth* parameter.

When *Bandstop Search* is activated the analyzer uses (or creates) the four markers *Mkr 1* to *Mkr 4* to locate the **bandstop region:**




- *Mkr 1* indicates the minimum of the peak.
- *Mkr 2* indicates the point on the left edge of the peak where the trace value is equal to the minimum plus *x dB Bandwidth.*
- *Mkr 3* indicates the point on the right edge of the peak where the trace value is equal to the minimum plus *x dB Bandwidth.*
- *Mkr 4* indicates the center of the peak, calculated as the arithmetic mean value of the *LBE* and *UBE* positions.

For a symmetric peak, the positions of *Max and Center* coincide.

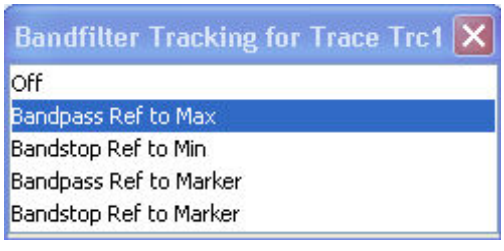
The band filter search results are displayed in the bandfilter info field.

**Remote control:** `CALCulate<Chn>:MARKer<Mk>:FUNCtion:BWIDth:MODE BSTOP`
`CALCulate<Chn>:MARKer<Mk>:FUNCtion:EXECute BFILter`

### Bandfilter Tracking

Causes the bandfilter search to be repeated after each sweep: When tracking mode is active the markers typically change their horizontal and their vertical positions as the measurement goes on.

Tracking for the different bandfilter search modes is enabled or disabled in a selection box. Selecting a search mode for tracking also activates this mode.

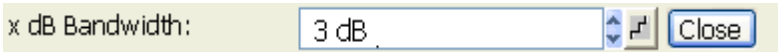



*Tracking* is a toggle function: Selecting the function repeatedly switches the tracking mode on and off.

**Remote control:** `CALCulate<Chn>:MARKer<Mk>:FUNCtion:EXECute BFILter`
`CALCulate<Chn>:MARKer<Mk>:SEARch:TRACking`

### x dB Bandwidth

Opens the numeric entry bar for the minimum excursion of the bandpass and bandstop peaks.

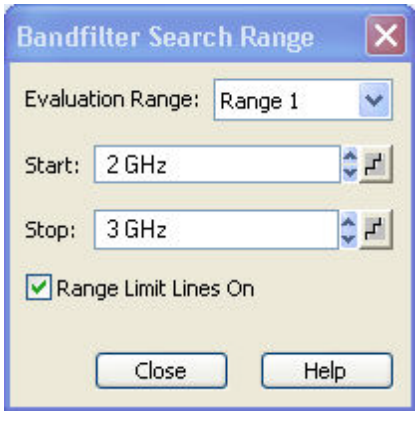



- A bandpass peak must fall off on both sides by the specified *x dB Bandwidth* to be considered a valid peak.
- A bandstop peak must increase on both sides by the specified *x dB Bandwidth* to be considered a valid peak.

**Remote control:** `CALCulate<Chn>:MARKer<Mk>:BWIDth <Level>`

## Search Range...

Opens the *Search Range Dialog* to confine the bandfilter search to a subrange of the sweep.




It is possible to define and store up to ten different search ranges for each trace. The bandfilter search is performed using the markers *Mkr 1, ..., Mkr 4,* irrespective of the selected search range.

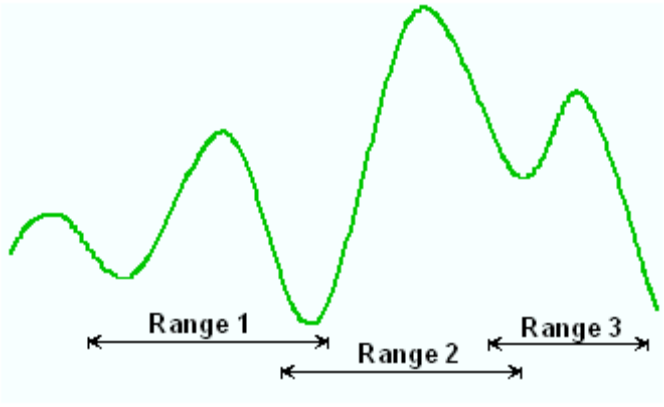
- *Search Range* Selects a search range for the bandfilter search. *Full Span* means that the search range is equal to the sweep range. Besides, it is possible to store up to 10 customized search ranges.
- *Start* defines the beginning of the search range. *Start* must be smaller than the *Stop* value, otherwise the search will not be initiated.
- *Stop* defines the end of the search range. *Stop* must be larger than the *Start* value, otherwise the search will not be initiated.
- *Range Limit Lines On* displays two vertical lines indicating the *Start* and the *Stop* value of the current bandfilter search range in the diagram area. This function is enabled as soon as one of the search ranges 1 to 10 is selected.

**Search range properties**

In contrast to the marker properties defined in the *Marker* and *Search* menus, the ten search ranges are valid for the entire setup. This means that, once defined, each of them can be used for any trace in the setup, irrespective of the channel that the trace belongs to.

The default search range of each new marker is *Full Span.* The analyzer provides greatest flexibility in defining search ranges. In particular, two search ranges may 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 ranges can be used to search a trace for several bandpass regions.




**Remote control:** `CALCulate<Chn>:MARKer<Mk>:FUNCtion:DOMain:USER`
`CALCulate<Chn>:MARKer<Mk>:FUNCtion:DOMain:USER:STARt`
`CALCulate<Chn>:MARKer<Mk>:FUNCtion:DOMain:USER:STOP`

## Search Result Off

Hides the info field with the results of a bandpass or a bandstop search and disables bandfiter *Tracking*. The info field is displayed again (and tracking re-enabled) when a new bandfilter search is performed.

| Bandpass Ref to Max Parameters | tracking... |
|---|---|
| Bandwidth: | 932.103853 MHz |
| Center: | 1.813672673 GHz |
| Lower Band Edge: | 1.347620747 GHz |
| Upper Band Edge: | 2.279724600 GHz |
| Quality Factor: | 1.9458 U |
| Loss: | 5.604 dB |

**Bandfilter parameters**

The info field contains the following search results:

- *Bandwidth* is the n-dB bandwidth of the bandpass/bandstop region, where n is the selected *x dB Bandwidth.* The bandwidth is equal to the difference between the Upper Bandwidth Edge (UBE) and the Lower Bandwidth Edge (LBE).
- *Center* is the stimulus frequency where the trace reaches its absolute maximum (minimum) within the bandpass (bandstop) region.
- *Lower Band Edge* is the closest frequency below the center frequency where the trace is equal to the center value minus n dB.
- *Upper Band Edge* is the closest frequency above the center frequency where the trace is equal to the center value minus n dB.
- The *Quality Factor* is the ratio between the *Center* frequency and the 3-dB *Bandwidth;* it does not depend on the selected *x dB Bandwidth.*
- *Loss* is the loss of the filter at its center frequency and is equal to the response value of marker no. 4. For an ideal bandpass filter the loss is zero (0 dB), for an ideal bandstop filter it is – ∞ dB.

**Remote control:** `CALCulate<Chn>:MARKer<Mk>:BWIDth?`
`CALCulate<Chn>:MARKer<Mk>SEARch:BFILter:RESult[:STATe]`

### Bandpass Search Ref to Marker

Activates the search for a bandpass region on the active trace and activates bandfilter *Tracking*, starting at the position of the active marker. A bandpass region is the closest peak in the search range with a minimum excursion specified by means of the *x dB Bandwidth* parameter.

In contrast to a *Bandpass Search Ref to Max,* the *Bandpass Search Ref to Marker* does not change the position of the active markers. The band filter search results are displayed in the bandfilter info field.

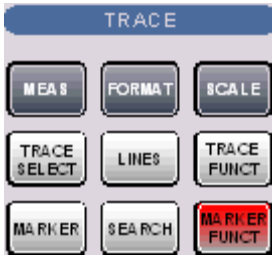
To search for a bandstop region in the vicinity of the active marker, use *Bandfilter Tracking – Bandstop Search Ref to Marker.*

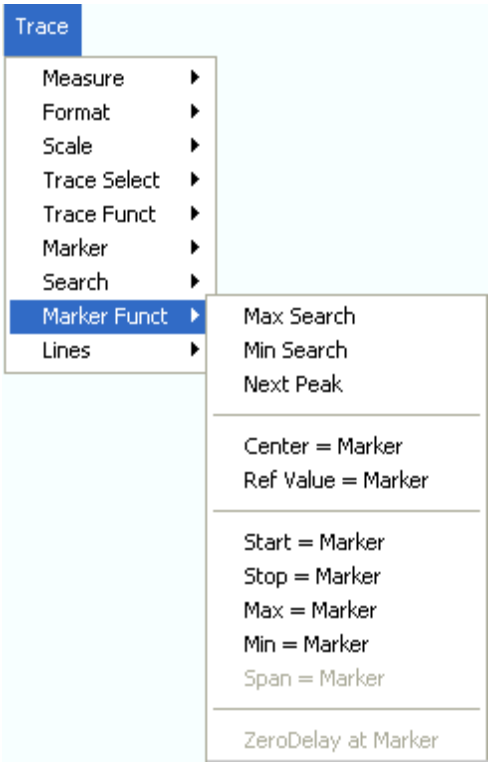
**Remote control:**

```
CALCulate<Chn>:MARKer<Mk>:FUNCtion:BWIDth:MODE BPRMarker | BSRMarker
CALCulate<Chn>:MARKer<Mk>:FUNCtion:EXECute BFILter
```

## Marker Funct

The *Marker Functions* use the active marker to define the sweep range, scale the diagram and introduce an electrical length offset.




The functions in the upper section are also provided in the *Search* submenu:

- *Max Search* sets the active marker to the absolute maximum in the search range.
- *Min Search* sets the active marker to the absolute minimum in the search range.
- *Next Peak* sets the active marker to the next maximum or minimum in the search range,

depending on the current search criterion.

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 marker value.
- *Start = Marker* sets the beginning of the sweep range equal to the marker value.
- *Stop = Marker* sets the end of the sweep range equal to the marker value.
- *Span = Marker* sets the span of the sweep range equal to the active delta marker value.

The following functions use the response value of the active marker to scale the y-axis of the diagram*:*

- *Ref. Value = Marker* sets the reference value equal to the marker value.
- *Max = Marker* sets the upper edge of the diagram equal to the marker value.
- *Min = Marker* sets the lower edge of the diagram equal to the marker value.
- *Zero Delay at Marker* corrects the measurement result by adding or subtracting a constant group delay.

## 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.

**Remote control:** `CALCulate<Chn>:MARKer<Mk>:FUNCtion:CENTer`

## 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.

**Remote control:** `CALCulate<Chn>:MARKer<Mk>:FUNCtion:STARt`

## 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.

**Remote control:** `CALCulate<Chn>:MARKer<Mk>:FUNCtion:STOP`

## Span = Marker

Sets the span of the sweep range equal to the absolute value of the first coordinate of the active delta marker, i.e. to the difference between the delta marker and the reference marker positions. The function is available only if the active marker is in Delta Mode. The reference marker appears at the right, the delta marker at the left edge of the diagram or vice versa, depending on which of the two markers has the smaller stimulus value.

## Ref. Value = 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.

### Zero Delay at Marker

Corrects the measurement result by adding or subtracting a constant group delay. This function must be applied to a trace which is displayed in group delay format. The trace is shifted in vertical direction so that the delay at the marker position vanishes.

The delay represents the propagation time of the wave across the DUT, so this operation corresponds to a electrical length compensation, i.e. to a shift of the reference plane by adding to or subtracting from the test port a simulated lossless transmission line of variable length. The correction 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).

*The Zero Delay at Marker function modifies the Offset parameters and therefore influences the entire channel.*

## Trace - Lines

The commands in the *Lines* define limits for the measurement results, visualize them in the diagrams and activate/deactivate the limit check. Besides the menu provides a horizontal line for each trace.

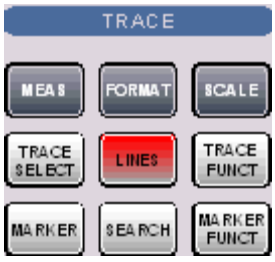
**Limit Lines**

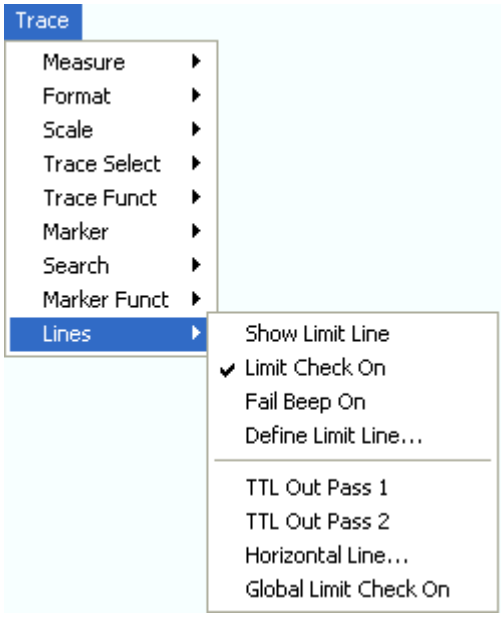
A limit line is a set of data to specify the allowed range for some or all points of a trace. Typically, limit lines are used to check whether a DUT conforms to the rated specifications (conformance testing).

- The *upper* limit line defines the maximum value for the trace points.
- The *lower* limit line defines the minimum value for the trace points.

A limit check consists of comparing the measurement results to the limit lines and display a pass/fail indication. An acoustic warning and a TTL signal at the *USER CONTROL* port can be generated in addition if a limit is exceeded.

Upper and lower limit lines are both defined as a combination of segments with a linear dependence between the measured quantity and the sweep variable (stimulus variable). The limit lines can be stored to a file and recalled. Data or memory traces can be used to define the segments of a limit line. Moreover it is possible to modify the limit lines globally by adding an offset to the stimulus or response values.



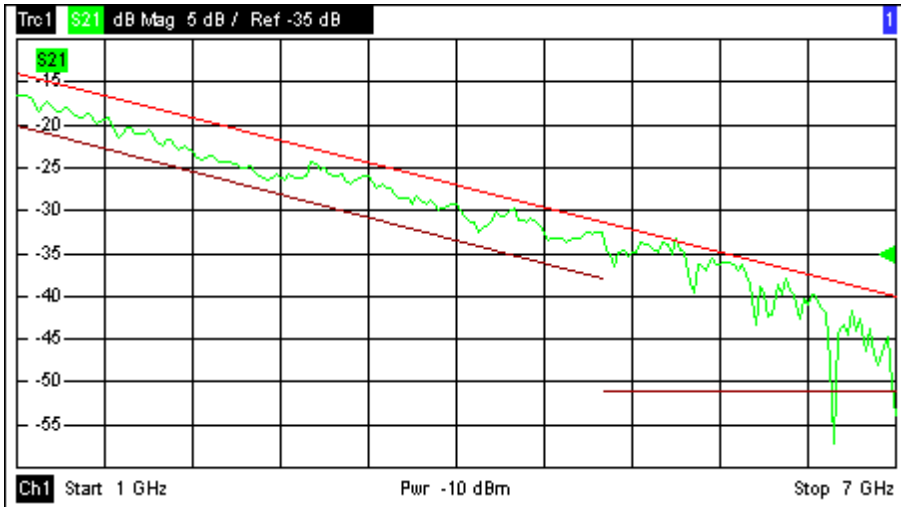

- *Show Limit Line* displays or hides the limit line associated to the active trace.
- *Limit Check On* activates or deactivates the limit check.
- *Fail Beep On* activates or deactivates the acoustic signal indicating a limit excess.
- *Define Limit Line* opens a dialog to define, save or recall limit lines.
- *TTL Out Pass 1/2* activates or deactivates a TTL output signal at the *USER CONTROL* connector indicating whether the trace was passed or failed.
- *Horizontal Line* displays or hides the horizontal line of the active trace and changes its position.
- *Global Limit Check* activates a global (composite) limit check on all traces in the active setup.

Limit lines are available for all Cartesian diagram types (Trace – Format). For polar diagrams, the functions of the *Lines* submenu are grayed. The limit lines are hidden and the limit check is disabled when a Cartesian trace format is replaced by a polar diagram.

## Show Limit Line

Shows or hides the limit line associated to the active trace in a Cartesian diagram area. A checkmark appears next to the menu item when the limit line is shown.

In the diagram, upper and lower limit lines can be displayed with different colors. Limit line segments with disabled limit check (see *Define Limit Line)* can also be colored differently. The limit line colors are defined in the *Define User Color Scheme* dialog *(Display – Display Config. – Color Scheme...).*



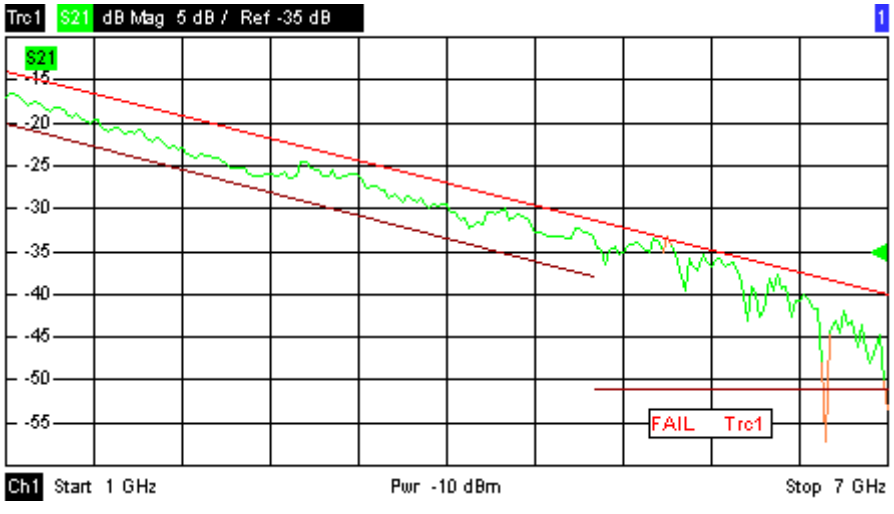

NOTE Display of the limit line and limit check are independent of each other: Hiding the limit line does not switch off the limit check.

**Remote control:** `CALCulate<Chn>:LIMit:DISPlay[:STATe] ON | OFF`

## Limit Check On

Switches the limit check of the active trace on or off. A checkmark appears next to the menu item when the limit check is enabled.

When the limit check is switched on, a *PASS* or *FAIL* message is displayed in the center of the diagram. If the limit check fails at a measurement point, the two trace segments to the left and right of the point can change their color. The *Limit Fail Trace* color is defined in the *Define User Color Scheme* dialog *(Display – Display Config. – Color Scheme...).* An acoustic signal (*Fail Beep*) and a TTL signal indicating pass or fail can be generated in addition.



*Limit check and display of the limit lines are independent of each other: With disabled limit check, the limit line can still be displayed, however, no display elements indicating a limit excess will appear.*

*Limits are checked at the actual measurement points, whereas a limit failure is indicated for the trace segments on both sides of a failed point. A small number of points causes wide trace segments so that the out-of tolerance regions can appear wider as they are.*

*If no limit lines are defined for the active trace, the limit check can be switched on but will always PASS the trace.*

**Remote control:**
```
CALCulate<Chn>:LIMit:STATe ON | OFF
CALCulate<Chn>:LIMit:LOWer:STATe ON | OFF
CALCulate<Chn>:LIMit:UPPer:STATe ON | OFF
CALCulate<Chn>:LIMit:FAIL?
```

## Fail Beep On

Activates or deactivates the fail beep. The fail beep is a low-tone acoustic signal that is generated each time the analyzer detects an exceeded limit. No fail beep can be generated if the limit check is switched off. A checkmark appears next to the menu item when the fail beep is enabled.

**Remote control:**
```
CALCulate<Chn>:LIMit:SOUNd:STATe ON | OFF
```

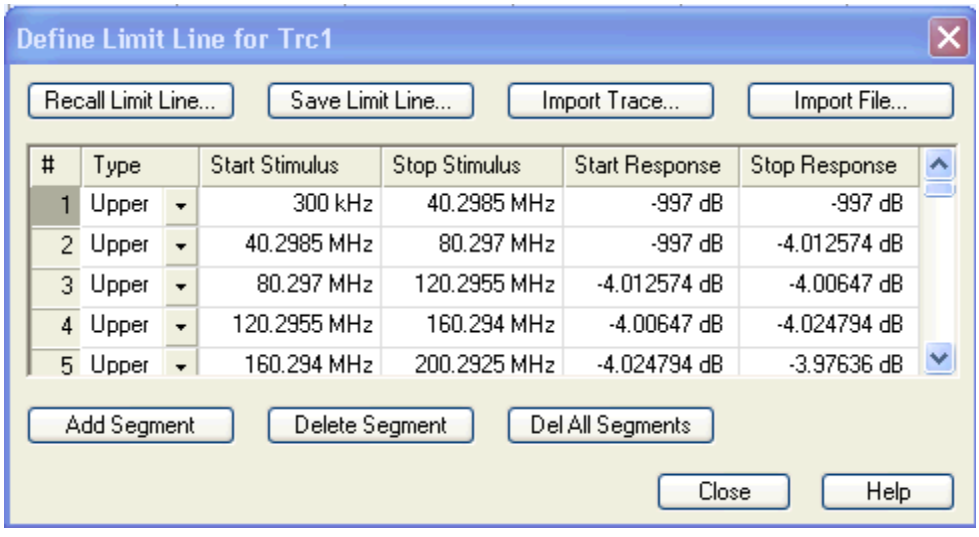
## Define Limit Line

Opens a dialog to define the limit line for the active trace on a segment-by-segment basis. In each segment the limit line is defined as a straight line connecting two points.

**Creating limit lines with minimum effort**

Choose one of the following methods to efficiently create and handle limit lines:

- To define a limit line with only a few segments, use *Add Segment* and edit each segment in the segment table individually.
- Use the multiselection feature to edit several limit line segments at the same time.
- 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 re-use or modify them later sessions *(Save Limit Line, Recall Limit Line)*.




The *Define Limit Line* dialog contains a table to edit the individual segments of the limit line; see below. The active trace is indicated in the title bar of the dialog. The three buttons below the table extend or shorten the segment list.

- *Add Segment* adds a new segment to the list. The new segment is a copy of the previously active segment and inserted after that segment. The current segment numbers are adapted. The analyzer places no restriction on the number of segments in a limit line.
- *Del. Sel. Segment* removes the selected segment from the list.
- *Del All Segments* clears the entire segment list so it is possible to define or load a new limit line.

The buttons to the right of the table are used to import and export limit line data.

- *Recall Limit Line...* calls up an *Open File* dialog to load a limit line from a limit line file. Limit line files are ASCII files with the default extension *.limit and a special file format.
- *Save Limit Line...* calls up a *Save As...* dialog to store the current limit line configuration to a limit line file. Limit line files are ASCII files with the default extension *.limit and a special file format.
- ***Import Trace***

  opens a box to select a trace that can be used to define a limit line.

  The box contains all data or memory traces of the active channel. As soon as a trace is selected, the *Properties of Imported Segments* dialogwith further global import options is opened.

- ***Import File***

  Calls up a standard *Import File* dialog to load a limit line from a trace file. The limit line import is analogous to the import of traces. Trace files are ASCII files with selectable file format. After the trace file is selected, the *Properties of Imported Segments* dialogwith further global import options is opened.

Imported traces are polygonal curves with n points and n – 1 segments. The number of points n is set via *Channel – Sweep – Number of Points*. The n – 1 segments are appended to the current segment

table for further editing. Existing limit line segments are not overwritten.

To import a limit line file (*.limit) you can also use the Windows Explorer and simply double-click the file or drag and drop the file into the NWA application. You have to switch on the limit check separately. Use the paste marker list for convenient entry of *Start* and *Stop* values.

**Columns** in the segment table

The table contains an automatically assigned current number for each segment plus the following editable columns:

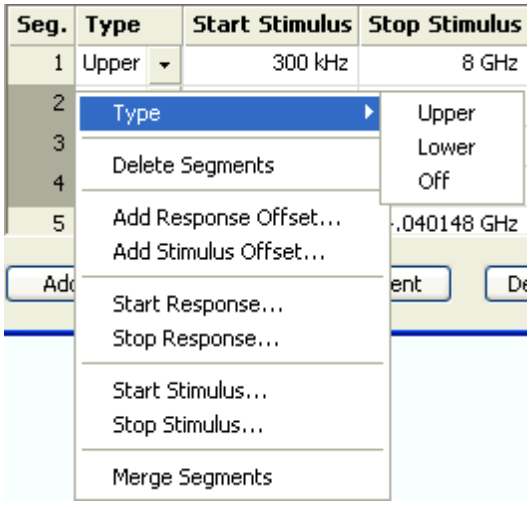
- *Type* indicates whether the segment belongs to an *Upper* or a *Lower* limit line, or if the limit check at the segment is switched *Off.* Switching off the limit check does not delete the segment but changes its screen color.
- *Start Stimulus* is the stimulus (x-axis) value of the first point of the segment (not necessarily smaller than *Stop Stimulus).*
- *Stop Stimulus* is the stimulus (x-axis) value of the last point of the segment (not necessarily larger than *Start Stimulus).*
- *Start Response* is the response (y-axis) value of the first point of the segment.
- *Stop Response* is the response (y-axis) value of the last point of the segment.

The limit line segment is calculated as a straight line connecting the two points (<Start Stimulus>, <Start Response>) and (<Stop Stimulus>, <Stop Response>); see *Rules for Limit Line Definition*.

**Remote control:**
```
CALCulate<Chn>:LIMit:CONTrol[:DATA]
CALCulate<Chn>:LIMit:DATA
CALCulate<Chn>:LIMit:SEGment<Seg>...
CALCulate<Chn>:LIMit:UPPer...
CALCulate<Chn>:LIMit:LOWer...
CALCulate<Chn>:LIMit:DELete:ALL
MMEMory:STORe:LIMit
MMEMory:LOAD:LIMit
```

## Multi-Selection of Limit Line Segments

In the *Define Limit Line* dialog it is possible to edit several limit line segments at the same time. Selection of two or more segments (use the left mouse key and the *Shift* key of an external keyboard) and a right-click on the dark grey *Seg.* area opens a context menu:




The context menu provides the following functions:

- Modification of all entries in the segment table: Type, start and stop values for the stimulus and response variable.
- Definition of an offset for response and stimulus values in analogy to the *Properties of Imported Segments* dialog.
- *Delete* the selected segments.
- *Merge* the selected segments to a single new segment. The start and stop values of the new segment are given by the start values of the first selected segment and the stop value of the last selected segment. The type is taken from the first selected segment. The new segment replaces the selected segments.

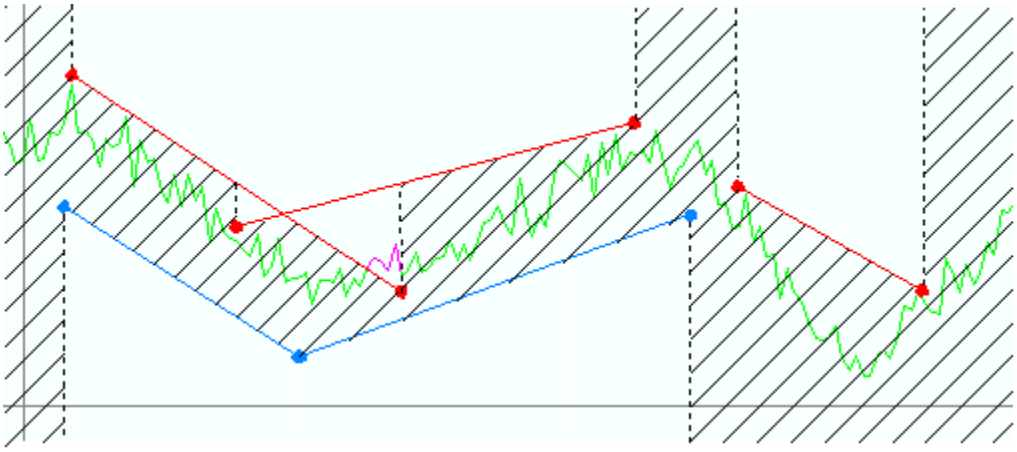
**Remote control:** `CALCulate<Chn>:LIMit:UPPer...`
`CALCulate<Chn>:LIMit:LOWer...`

## Rules for Limit Line Definition

The analyzer places very few restrictions on the definition of limit line segments. The following rules ensure a maximum of flexibility:

- Segments don't have to be sorted in ascending or descending order (e.g. the *Start Stimulus* value of segment no. n doesn't 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 refers 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.

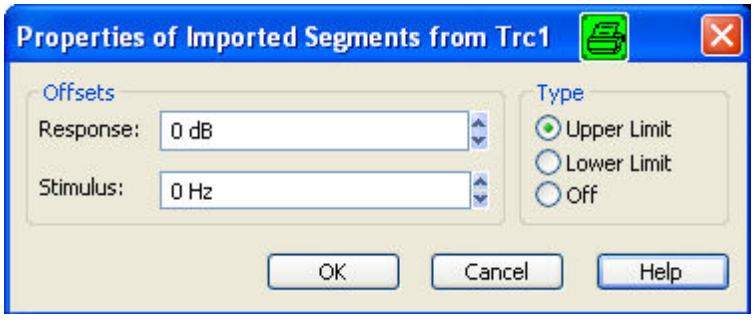
The following figure shows a limit line consisting of 3 upper and 2 lower limit line segments. To pass the limit check, the trace must be confined to the shaded area.



As a consequence of the limit line rules, the limit check will always pass a DUT if no limit lines are defined.

## File Import Settings

The *Properties of Imported Segments* dialog appears before a trace is imported into the *Define Limit Line* dialog.




The dialog assigns common properties to all limit line segments generated by the imported trace.

- *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 control:**

```
CALCulate<Chn>:LIMit:LOWer:FEED
<stimulus_offset>,<response_offset>[,<trace_name>]
CALCulate<Chn>:LIMit:UPPer:FEED
<stimulus_offset>,<response_offset>[,<trace_name>]
MMEMory:LOAD:LIMit
```

## 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 *Export Limit Line* and *Import 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:

| Seg. | Type | Start Stimulus | Stop Stimulus | Start Response | Stop Response |
|---|---|---|---|---|---|
| 1 | Upper | 1 GHz | 3 GHz | 2 dB | 2 dB |
| 2 | Lower | 3 GHz | 8 GHz | 3 dB | 4 dB |
| 3 | Off | 300 kHz | 1 GHz | 1 dB | 1 dB |

is described by the limit line file:

```
# version 1.00
#
# Channel 2
# Trace 2
en:Type; Start Stimulus[MHz];    Stop Stimulus[MHz];     Start Response[dB];     Stop Response[dB];
Upper;   1.000000000000000E+003; 3.000000000000000E+003; 2.000000000000000E+000; 2.000000000000000E+000;
Lower;   3.000000000000000E-006; 8.000000000000000E+003; 3.000000000000000E+000; 4.000000000000000E+000;
Off;     3.000000000000000E-001; 1.000000000000000E+003; 1.000000000000000E+000; 1.000000000000000E+000;
```

**Remote control:** `MMEMory:LOAD:LIMit "Trc_name","file_name"`
`MMEMory:STORe:LIMit "Trc_name","file_name"`

## TTL Out Pass 1/2

Assigns the active trace to the low-voltage (3.3 V) TTL output signals at the *USER CONTROL* connector. To select *TTL Out Pass 1 / 2*, the limit check of the active trace must be switched on.

- If *TTL Out Pass 1* is selected and the active trace is within limits, then the TTL signal is applied to pin 13 of the *USER CONTROL* connector.
- If *TTL Out Pass 2* is selected and the active trace is within limits, then the TTL signal is applied to pin 14 of the *USER CONTROL* connector.

If the active trace exceeds the limits, then no TTL signal is generated. It is possible to activate both pass/fail signals for the same trace or assign several traces to a signal.

**Extension: Monitoring several traces**

If a channel contains several traces, is possible to assign them one after another to each pass/fail signal. The procedure divides the traces of the channel into four groups that are either assigned to signal A, to signal B, to both signals, or to none of them.

If several traces with independent limit check are assigned to a pass/fail signal, then the TTL signal is generated only if all traces are within limits. It is switched off as soon as one trace exceeds the limits.

**Application: Graduated limit check**

The two pass/fail signals can be used to distinguish three quality levels of a DUT. The test is performed with a looser and a tighter set of limit lines that are assigned to two traces with identical channel and trace settings. The limit check for the two traces is monitored by means of the signals *TTL Out Pass 1 /TTL Out Pass 2,* respectively.

- If the DUT is passed in both limit checks, the quality is good.
- If the DUT is failed in the limit check with tighter limits, the quality is still sufficient.
- If the DUT is failed in both limit checks, the quality is poor.

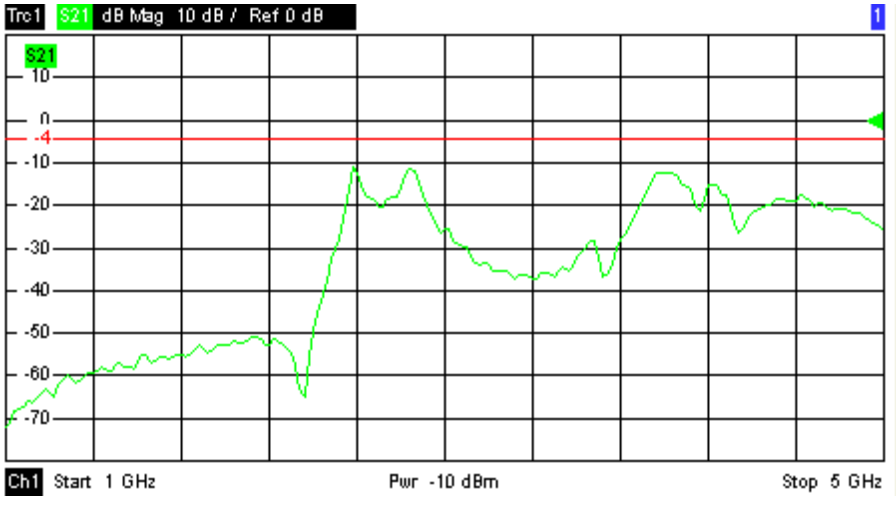
Instead of using two traces, it is possible to consider two groups of traces that are assigned to *TTL Out Pass 1* and *TTL Out Pass 2*.

**Remote control:** `CALCulate<Chn>:LIMit:TTLout<nr>[:STATe] ON | OFF`

## Horizontal Line

Shows or hides the horizontal line associated to the active trace in a Cartesian diagram area. A checkmark appears next to the menu item when the horizontal line is shown.

The horizontal line (or display line) is a red line which can be moved to particular trace points in order to retrieve the response values.



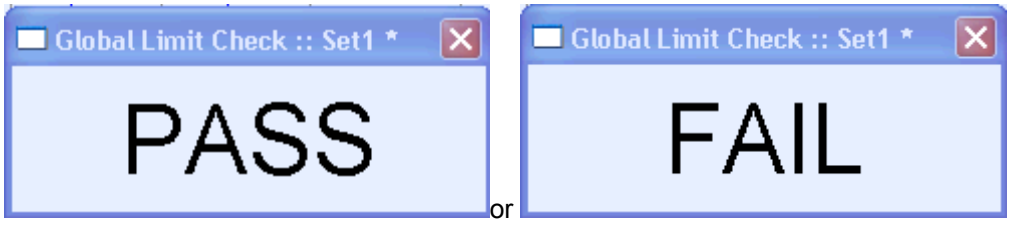

- Pressing *Horizontal Line* for a first time shows the line for the active trace and opens the numeric entry bar to define its position (response value). The (rounded) position is displayed near the left edge of the screen.
- Pressing *Horizontal Line* for a second time hides the horizontal line for the active trace.

TIP Use the analyzer's drag-and-drop functionality to move the horizontal line symbol to the desired position.

**Remote control:** `CALCulate<Chn>:DLINe:STATe ON | OFF`
`CALCulate<Chn>:DLINe`

## Global Limit Check

Performs a composite limit check on all traces of the current setup. The result of the global check appears in a popup box whenever *Global Limit Check* is pressed.



or

- *PASS* represents pass for all traces for which the limit check is enabled. A trace without limit lines or with disabled individual limit check always passes the composite limit check.
- *FAIL* means that the limit checks for one or more traces failed.

**Remote control:** `CALCulate<Chn>:CLIMits:FAIL?`

# Channel Menu

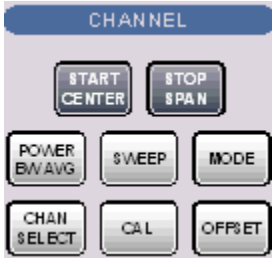
The *Channel* menu provides all channel settings and the functions to activate, modify and store different channels.

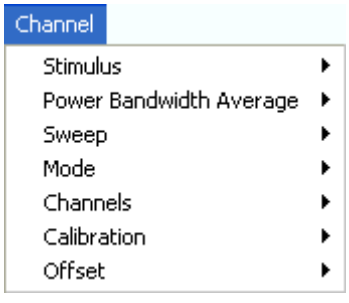
**Channels**

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

- Control of the measurement process *(Sweep, Trigger, Average)*
- Description of the test setup *(internal source power, IF filters and step attenuators)*
- Correction data *(Calibration, Offset)*

The channel settings complement the definitions of the *Trace* menu. Each trace is assigned to a channel, see Traces, Channels and Diagram Areas. The channel settings apply to all traces assigned to the channel.



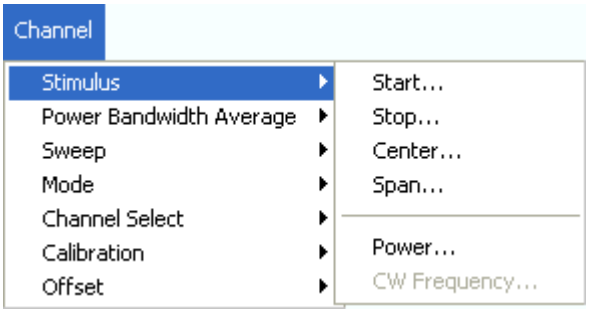

The *Channel* menu contains the following functions and submenus:

- *Stimulus* defines the sweep range, depending on the sweep type.
- *Power Bandwidth Average* defines the power of the internal signal source, sets the step attenuators and the IF bandwidths, and configures the sweep average.
- *Sweep* defines the scope of measurement, including the sweep type, the trigger conditions and the periodicity of the measurement.
- *Mode* opens the *Port Configuration* dialog to define the properties of the physical and logical (balanced) test ports.
- *Channel Select* provides functions to handle and activate channels.
- *Calibration* provides all functions that are necessary to perform a system error correction (calibration).
- *Offset* provides a selection of length offset parameters to shift the measurement plane.

## Stimulus

The *Stimulus* submenu defines the sweep range in the current channel, depending on the sweep type.

- In Cartesian diagrams, the sweep range corresponds to the diagram width and defines the scaling of the x-axis.
- In polar diagrams and Smith charts the stimulus axis is lost but the marker functions easily provide the stimulus value of any measurement point.

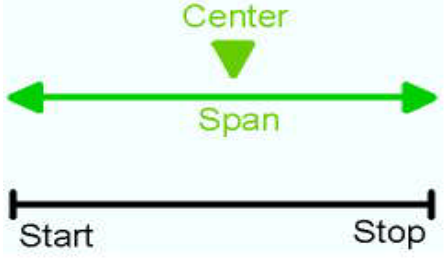



- *Start* is the lowest value of the sweep variable (e.g. the lowest frequency measured) and corresponds to the left edge of the Cartesian diagram.
- *Stop* is the highest value of the sweep variable (e.g. the highest frequency measured) and corresponds to the right edge of the Cartesian diagram.
- *Center* corresponds to the center of the Cartesian diagram, i.e. (Start + Stop)/2.
- *Span* corresponds to the diagram width, i.e. (Stop – Start).
- *Power* defines the power of the internal signal source.
- *CW Frequency* sets the fixed frequency for the sweep types *Power, Time* and *CW Mode.*
- *START CENTER* opens the input field for the *Start* frequency, if the range is currently defined by *Start* and *Stop.* It opens the input field for the *Center* frequency, if the range is currently defined by *Center* and *Span.*
- *STOP SPAN* opens the input field for the *Stop* frequency, if the range is currently defined by *Start* and *Stop.* It opens the input field for the *Span*, if the range is currently defined by *Center* and *Span.*

The Marker Functions provide a convenient alternative to the manual entry of the sweep range parameters. Use the paste marker list for convenient entry of *Start* and *Stop* values.

Some of the *Stimulus* parameters and the *STOP SPAN* key are not provided for all sweep types; see Stimulus and Sweep Types.

**Equivalence of settings:** For a frequency sweep the *Start* and *Stop* frequencies or the *Center* frequency and *Span* are alternative settings*:*




**Remote control:**

```
[SENSe<Ch>:]FREQuency:STARt
[SENSe<Ch>:]FREQuency:STOP
[SENSe<Ch>:]FREQuency:CENTer
[SENSe<Ch>:]FREQuency:SPAN
```

## Stimulus and Sweep Types

The function and input values of the *Start, Stop, Center* and *Span* softkeys depend on the sweep type:

| Sweep type | Start [dimension] | Stop [dimension] | Center [dimension] | Span [dimension] |
|---|---|---|---|---|
| Equivalent hotkey | START CENTER | STOP SPAN | START CENTER (press twice) | STOP SPAN (press twice) |
| **Lin. Frequency**<br>**Log. Frequency**<br>**Segmented Frequency** | Start frequency [Hz] | Stop frequency [Hz] | Center frequency [Hz] | Frequency span [Hz] |
| **Power** | Start power [dBm] | Stop power [dBm] | – | – |
| **Time** | – | Total sweep time [s] | – | – |
| **CW Mode** | – | Number of points [1] | – | – |

The ranges of numerical values must be compatible with the instrument model.

If a frequency sweep is selected, then the *START CENTER* and *STOP SPAN* are both toggle keys. In *Time* or *CW Mode* sweep mode the *Start, Center* and *Stop* softkeys are inactive, still you can use the *START CENTER* hardkey to select the fixed CW frequency.

The conditions for the stimulus range depend on the sweep type:

- **Lin. Frequency/Log. Frequency/Segmented Frequency**

  The frequency range depends on the instrument type; see SCPI command reference. The *Stop* frequency must be greater than the *Start* frequency; the *Span* must be ≥ 1 Hz. If a *Stop* frequency smaller than the current *Start* frequency is set, then the *Start* frequency is adjusted and vice versa).

  A sweep must contain at least two different sweep points.

- **Power**

  *Start* and *Stop* power are entered in absolute units (dBm) and over a wide range. The *Center* and *Span* softkeys are not active.

  *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 isset, then the *Start* power is adjusted automatically and vice versa.

  The power corresponds to the actual output power at the test port that supplies the stimulus for the active channel or at the calibrated reference plane.

- **Time**

  The *Stop* value represents the total sweep time. *Start* and the other parameters are not active as the sweep time is relative to the beginning of the sweep which starts immediately after the entry of the *Stop* value.

  The sweep time is entered in seconds and must be positive; see also Time sweep.

- **CW Mode**

  The *Stop* value represents the number of trigger events during the

measurement. Each trigger event triggers one measurement point. *Start* and the other softkeys are not active as the first trigger event is the one immediately after the beginning of the sweep which starts after the entry of the *Stop* value.

The behavior is thus analogous to the *Time* sweep type.

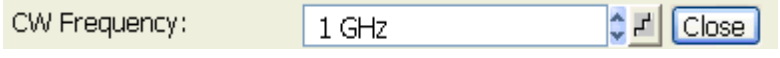
- **Pulse Profile**

  *Start* and *Stop* define the start and stop time in the diagram area relative to the trigger time. The stop time must be larger than the start time. *Center* and *Span* are not active.

The selected sweep range applies to all source and receiver ports of the analyzer. The sweep range frequency is also used as the channel base frequency for frequency-converting measurements; see *Port Configuration.*

### CW Frequency

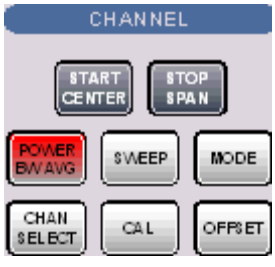
Opens the numeric entry bar to set the fixed frequency for the sweep types *Power, Time, CW Mode,* and *Pulse Profile*. The CW frequency is also used as the channel base frequency for frequency-converting measurements; see *Port Configuration.*

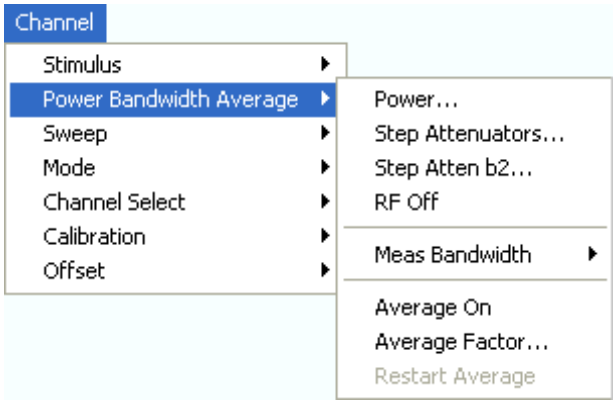



**Remote control:** `SOURce<Ch>:FREQuency<Pt>:CW|FIXed`
`[SENSe<Ch>:]SWEep:CW|FIXed`

## Power Bandwidth Average

The *Power Bandwidth Average* menu defines the power of the internal signal source, sets the step attenuators and the IF bandwidths, and configures the sweep average.



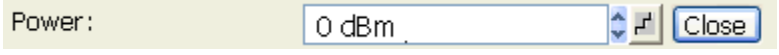

- *Power* defines the power of the internal signal source.
- *Step Attenuators* sets the attenuation for all received and generated waves.
- *Step Atten. b2...*sets the attenuation for the received wave b2.
- *RF Off* switches the internal and external power sources on (if checked) or off.

- *Meas. Bandwidth* selects the bandwidth of the IF measurement filter.
- *Average On* activates or de-activates the sweep average. With average on the measurement results are averaged over a selected number of consecutive sweeps (Average Factor).
- *Average Factor* defines the number of consecutive sweeps to be averaged.
- *Restart Average* starts a new average cycle, clearing all previous results and thus eliminating their effect on the new cycle. The new cycle is started as fast as possible; an ongoing sweep is terminated immediately.

**Remote control (for RF off and averaging):**
```
DIAGnostic:SERVice:RFPower
[SENSe<Ch>:]AVERage[:STATe] ON | OFF
[SENSe<Ch>:]AVERage:COUNt
[SENSe<Ch>:]AVERage:CLEar
```

## Power

Opens the numeric entry bar to set the power of the internal signal source (channel power).




The channel power determines the output power at the test ports if a *Frequency Sweep* or a *Time Sweep* is active; see info below. The setting has no effect in *Power Sweep* mode, where the source power is varied over a continuous range.

The selected channel power applies to all source ports used in the active channel. Use the *Port Configuration* dialog to modify the output power ranges for the individual analyzer ports.

**Output power at the test ports**

*Power* sets the output power at the test port that supplies the stimulus for the active channel. The channel power can be varied over a wide dynamic range; the range can be further increased by means of generator step attenuators. This leaves enough flexibility to include an attenuation or gain in the test setup.

The channel power can be modified by the following settings:

- Generator step attenuators decrease the output power by a definite factor $a_{step}$. $a_{step}$ can be set individually for all test ports but should not change during a sweep. To ensure a sufficient dynamic range at the test ports, it is recommended to use automatic generator step attenuator setting.
- The Source section of the *Port Configuration* dialog provides a port-specific constant power offset (for arbitrary sweeps) and a port and frequency-dependent power slope factor (for frequency sweeps).
- A generator (source) power calibration generally modifies the channel power so that *Power* is equal to the power at the calibrated reference plane.

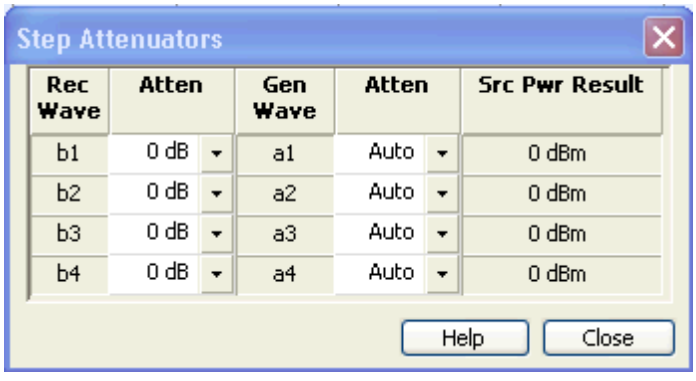
The *Port <port_no> Source Power* dialog gives an overview of all power settings.

**Remote control:**
```
SOURce<Ch>:POWer[:LEVel][:IMMediate][:AMPlitude]
```

## Step Attenuators (R&S ZVA and R&S ZVB Analyzers)

Opens a dialog to set the attenuation for the received and generated waves. The generator attenuation

can be set automatically; see *Automatic generator attenuation* below.




The Step Attenuators dialog contains the following columns:

- *Rec. Wave* lists the received waves $b_1$ to $b_n$. n is equal to the total number of test ports of the analyzer.
- *Attenuation* is used to enter the attenuation for each wave $b_n$.
- *Gen. Wave* lists the generated waves $a_1$ to $a_n$.
- *Attenuation* is used to enter the attenuation for each wave $a_n$. If *Auto* is selected, the analyzer automatically optimizes the generator attenuation according to the current channels settings, see background information below.
- *Src Pwr Result* displays the source power according to the current channel and generator attenuator settings.

Attenuation factors can be entered only for ports where a receiver or generator step attenuator is installed.

**Generator and receiver step attenuators**

Receiver and generator step attenuators can be purchased as ZVA options. The following attenuators are available:

| | Port 1 | Port 2 | Port 3 | Port 4 |
|---|---|---|---|---|
| ZVA8, generator | ZVA8-B21 | ZVA8-B22 | ZVA8-B23 | ZVA8-B24 |
| ZVA8, receiver | ZVA8-B31 | ZVA8-B32 | ZVA8-B33 | ZVA8-B34 |
| ZVA24, generator | ZVA24-B21 | ZVA24-B22 | ZVA24-B23 | ZVA24-B24 |
| ZVA24, receiver | ZVA24-B31 | ZVA24-B32 | ZVA24-B33 | ZVA24-B34 |
| ZVA40, generator | ZVA40-B21 | ZVA40-B22 | ZVA40-B23 | ZVA40-B24 |
| ZVA40, receiver | ZVA40-B31 | ZVA40-B32 | ZVA40-B33 | ZVA40-B34 |

Refer to the data sheet for detailed ordering information and technical specifications.

Attenuation factors for the received waves and the generated waves serve different purposes:

- *Received wave factors* are used to adjust the received signal levels to the input level range of the analyzer in order to avoid damage to the instrument, e.g. if the DUT is a power amplifier. The range of attenuation values depends on the analyzer model.

Port-specific attenuation factors can be set in the *Receiver* section of the *Port Configuration* dialog.

- Generated wave factors are used to protect a sensitive DUT from excess input levels. The attenuation values can be set automatically. The range of values depends on the analyzer model.

  Port-specific attenuation factors can be set in the *Source* section of the *Port Configuration* dialog.

**Automatic generator attenuation**

The aim of the automaticattenuatorsetting is to ensure that the analyzer can perform the entire sweep without switching the generator step attenuators. The output power $p_{el}$ at each port depends on the following settings:

- The channel power $p_{ch}$ (the *Power* set in the *Channel – Power Bandwidth Average* menu).
- The port-specific power offset $p_b$ (for arbitrary sweeps) and the port and frequency-dependent power slope factor $p_{Slope}$ (for frequency sweeps) defined in the Receiver section of the *Port Configuration* dialog.
- A port and stimulus-dependent correction factor $p_{corr}$ introduced by a generator (source) power calibration.
- The port-specific generator step attenuator setting $a_{step}$.

If everything is expressed in dB units, the output power $p_{el}$ at each port must be equal to the sum of the channel power and all port and stimulus-dependent correction factors:

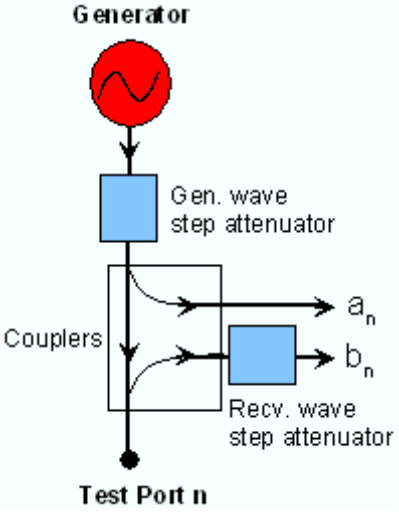
$$p_{el} = p_{ch} + p_b + p_{Slope} + p_{corr} + a_{step}$$

The automatic attenuator setting algorithm selects $a_{step}$ such that the variation range for $p_{el}$ is sufficient to account for the stimulus-dependent variations of $p_b$ and $p_{Slope}$, and $p_{corr}$. If this is not possible in a particular subrange of the sweep (e.g. because the sweep settings are such that an excess variation range for $p_{el}$ or a negative attenuation factor $a_{step}$ would be required) then the analyzer generates an error message.

NOTE

*A new system error correction is recommended whenever the step attenuator settings are changed. A change of the generator step attenuator settings may also occur in Auto mode; see above.*

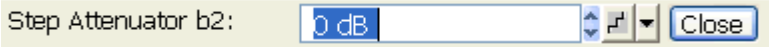
The position and function of the step attenuators is illustrated in the following figure. The analyzer measures both the attenuated generated waves and the attenuated waves received from the DUT.



**Remote control:**
```
INPut<Pt>:ATTenuation
[SENSe<Ch>:]POWer:ATTenuation ARECeiver | BRECeiver | CRECeiver
| DRECeiver, <attenuation>
OUTPut<Pt>:ATTenuation
SOURce<Ch>:POWer<Pt>:ATTenuation
SOURce<Ch>:POWer<Pt>:ATTenuation:AUTO ON | OFF
```

## Step Atten. b2 (R&S ZVA Analyzers)

Opens the numeric entry bar to set the attenuation for the received wave b2. This function is available only if a receiver step attenuator is installed at port 2.




The attenuation is used to adjust the received signal level at port 2 to the input level range of the analyzer in order to avoid damage to the instrument, e.g. if the DUT is a power amplifier. The range of values depends on the analyzer model.

Using a single step attenuation b2 corresponds to a standard test setup with 0 dB generator attenuation and attenuation of the (amplified) wave received at test port 2. The attenuation factors for all received and generated waves can be set via Step Attenuators.

**Remote control:**
```
INPut<Pt>:ATTenuation
[SENSe<Ch>:]POWer:ATTenuation ARECeiver | BRECeiver | CRECeiver
| DRECeiver, <attenuation>
```

## RF Off

*RF Off* switches the internal and external power sources on (if checked) or off. Switching off the RF power helps to prevent overheating of a connected DUT while not measurement results are taken.

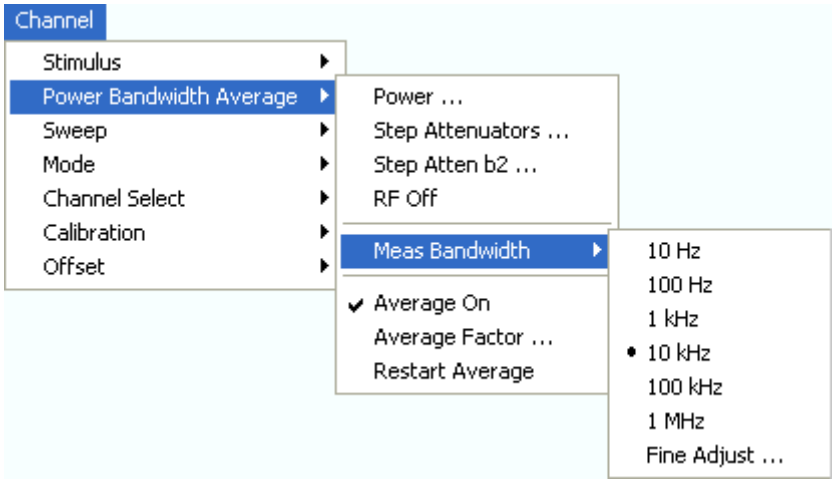
*After the RF power is switched on, the start of the measurement is delayed by approx. 1 s while the amplifier stages of the analyzer are allowed to settle. To circumvent this delay if you need to turn off the base channel power for a limited period of time, reduce the base channel power setting or turn off the drive ports in the Port Configuration dialog instead of using RF Off.*

Switching off the internal RF generator while an external generator is used can improve the measurement accuracy. *RF Off* also deactivates external generators, so you have to use the settings in the *Source* section of the *Port Configuration* dialog (with option R&S ZVA-K4).

**Remote control:** `OUTPut<Ch>[:STATe]`

## Meas. Bandwidth

Sets the measurement bandwidth of the IF filter. *Meas Bandwidth* opens a submenu to select bandwidths between 10 Hz and 1 MHz:




*Fine Adjust...* opens a dialog where you can modify the selected measurement bandwidth and the selectivity of the IF filter.

- *Bandwidth* shows the last IF filter bandwidth selected. The arrow buttons increment and decrement the bandwidth in 1-2-5 steps for each decade. Entered values between the steps will be rounded up, values exceeding the maximum bandwidth rounded down.
- *Selectivity* selects between two types of IF filters: Filters with *Normal* selectivity and short settling time, and filters with *High* selectivity but larger settling time.

The selected bandwidth and selectivity applies to all filters used in the current channel. This makes sense because the measurement speed is limited by the slowest filter in the channel. In *Segmented Frequency* sweeps, the bandwidth and selectivity can be set independently for each segment; see *Define Segments*.

**Optimizing the filter settings**

A high selectivity and a small filter bandwidth both suppress the noise level around the measurement frequency and thus increase the dynamic range. On the other hand the time needed to acquire a single measurement point increases for small 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 makes it particularly suitable for isolating unexpected spurious responses or known mixer products.

*In general, the system error correction is no longer valid after a change of the IF filter bandwidth. The message Cal? appears in the trace list.*

**Remote control:** `[SENSe<Ch>:]BANDwidth|BWIDth[:RESolution]`
`[SENSe<Ch>:]BANDwidth|BWIDth[:RESolution]:SELect`

### Average Factor

Opens the numeric entry bar to define the number of consecutive sweeps to be averaged.

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.

*The average factor is also valid for calibration sweeps: The calculation of system correction data is based on the averaged trace.*

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 in the vicinity of the measurement frequency, alternative techniques (e.g. a smaller filter bandwidth) must be used.

**Calculation of sweep average**

The average trace is obtained as follows:

Let c be the Average Factor and assume that n sweeps have been measured since the start of the average cycle (start of the measurement or *Restart Average)*. The following two situations are distinguished:

- n < c: At each sweep point, the average trace no. n is calculated from the average trace no. n – 1 and the current trace no. n according to the following recurrence:

  $$Avg(n) = \frac{n-1}{n} Avg(n-1) + \frac{1}{n} Curr(n) \qquad (n = 1, \ldots, c)$$

  The average trace represents the arithmetic mean value over all n sweeps.

- n > c: At each sweep point, the average trace no. n is calculated from the average trace no. n – 1 and the current trace no. n according to:

  $$Avg(n) = \frac{c-1}{c} Avg(n-1) + \frac{1}{c} Curr(n) \qquad (n > c)$$

The formulas hold for Average Factor n = 1 where the average trace becomes equal to the current trace.

**Remote control:**
```
[SENSe<Ch>:]AVERage[:STATe] ON | OFF
[SENSe<Ch>:]AVERage:COUNt
[SENSe<Ch>:]AVERage:CLEar
```

## Sweep

The *Sweep* submenu defines the scope of measurement in the current channel. This includes the sweep type with various parameters, the trigger conditions and the periodicity of the measurement.

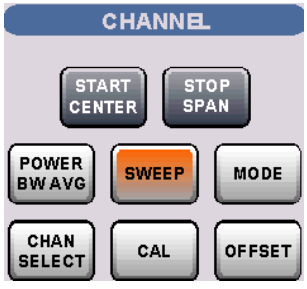
### Sweeps

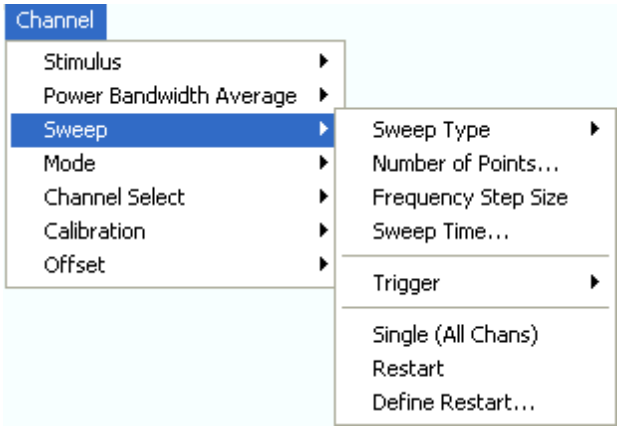
A sweep is a series of consecutive measurements taken over a specified sequence of stimulus values. It represents the basic measurement cycle of the analyzer.

The analyzer can perform sweeps at constant power but variable frequency (frequency sweeps), sweeps at constant frequency but variable power (power sweeps), and sweeps at constant power and frequency that are repeated in time (Time/CW Trigger Event sweeps); see Sweep Type.

The sweeps are further specified by the number of measurement points, the total measurement time and the trigger mode. A measurement may consist of a single sweep or a series of sweeps repeated continuously.

On the other hand, depending on the measurement task and the measured quantity, the measurement at each point can consist of several partial measurements with definite hardware settings.



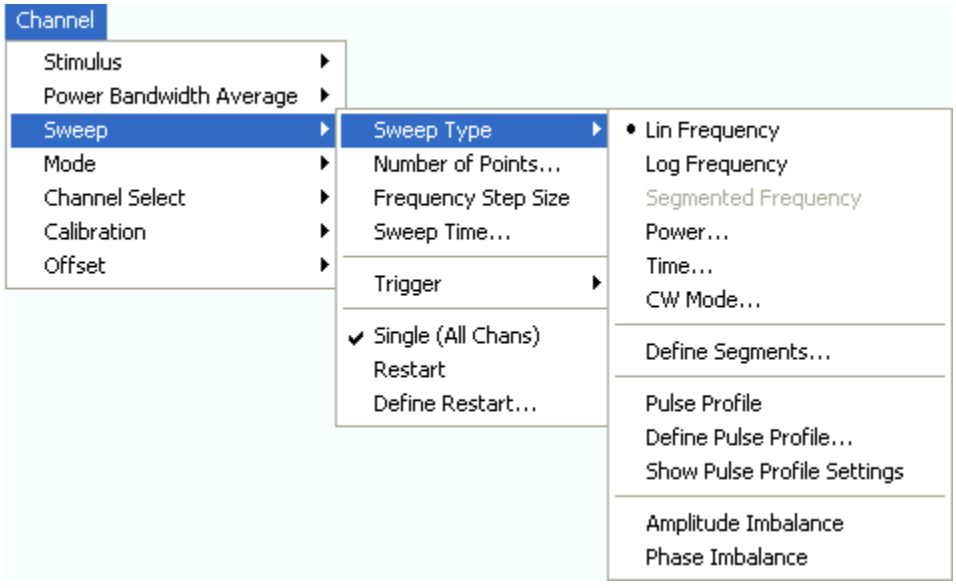




The *Sweep* menu provides the following settings:

- *Sweep Type* defines the sweep variable (frequency/power/time) and the position of sweep points in the sweep range.
- *Number of Points* sets the total number of measurement points over the sweep range.
- *Frequency Step Size* sets the distance between two consecutive frequency sweep points.
- *Sweep Time* varies the measurement time for a sweep or delays the start of each sweep.
- *Trigger* selects the trigger mode for the measurement.
- *Manual Trigger* while the *Manual* trigger mode is active generates a trigger event. This softkey is also available in the *Trigger* submenu.
- If *Single (All Chans)* is selected the measurement is terminated after a single sweep or a group of single sweeps defined in the *Define Restart* dialog.
- *Restart* aborts the current measurement and restarts a new sweep sequence.
- *Define Restart* opens a dialog to specify how many sweeps are repeated.

## Sweep Type

The *Sweep Type* submenu defines the sweep variable (frequency/power/time) and the position of the sweep points across the sweep range.

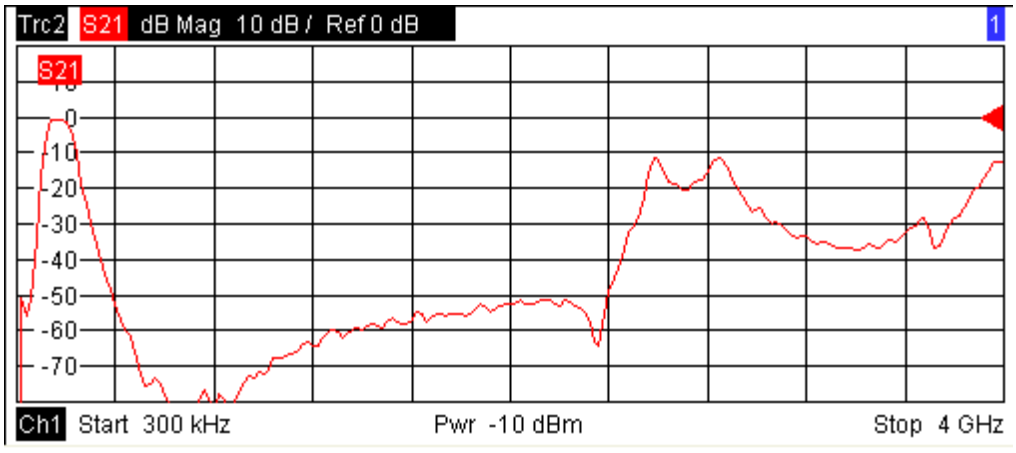



- *Lin. Frequency* is the default sweep type. The stimulus frequency is swept in equidistant steps over the continuous frequency range. In a Cartesian diagram, the x-axis is a linear frequency axis.
- *Log. Frequency* is analogous to *Lin. Frequency,* however, the frequency is swept in equidistant steps on a logarithmic scale. In a Cartesian diagram, the x-axis is a logarithmic frequency axis.
- *Segmented Frequency* is analogous to *Lin. Frequency* but uses a sweep range that can be composed of several continuous frequency ranges or single frequency points defined via Define Segments. A valid segment list must be defined before activating *Segment Frequency.*
- *Power* sweeps are performed at constant frequency but with variable generator power that is swept in linear, equidistant steps over a continuous range. In a Cartesian diagram, the x-axis is a dB-linear power axis.
- *Time* sweeps are performed at constant frequency and stimulus power. The measurement is repeated over a specified period of time at constant time intervals.
- *CW Mode* is a variant of the *Time* sweep where the individual measurements are triggered according to the current trigger settings.
- *Pulse Profile, Define Pulse Profile* and *Show Pulse Profile Settings* control the pulsed sweep mode (with option R&S ZVA-K7).
- *Amplitude Imbalance* and *Phase Imbalance* activate special sweep types for the true differential mode (with option R&S ZVA-K6).

## Lin. Frequency

In a *Lin. Frequency* sweep the stimulus frequency is swept in equidistant steps over the continuous frequency range. The frequency range (sweep range) is defined with the Stimulus settings. 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). The internal generator power can be set, if so desired, in the Power Bandwidth Average submenu.

A *Lin. Frequency* sweep corresponds to the analysis of a signal over the frequency, as obtained e.g. by means of a spectrum analyzer. This is the default sweep type. In a Cartesian diagram the measurement result is displayed as a trace over a linear frequency scale (spectral representation). The following example shows a *Lin. Frequency* sweep with a stimulus range between 4.5 GHz and 6 GHz, the forward transmission parameter S12 as measured quantity, and a *dB Mag* scaled y-axis.

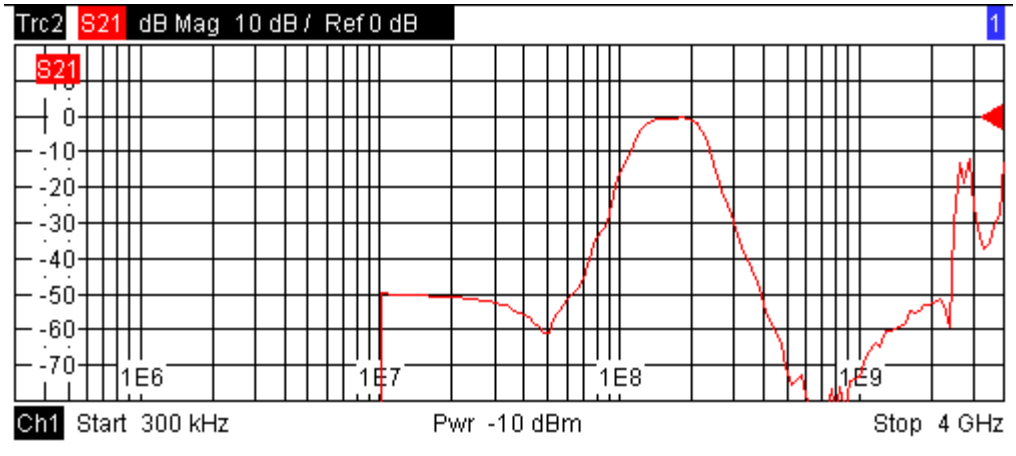



**Remote control:** `[SENSe<Ch>:]SWEep:TPYE LINear`
`[SENSe<Chn>:]FUNCtion[:ON] "XFRequency:..."`

## Log. Frequency

In a *Log. Frequency* sweep the stimulus frequency is swept on a logarithmic scale over the continuous frequency range. The frequency range (sweep range) is defined with the Stimulus settings. 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. The internal generator power can be set, if so desired, in the Power Bandwidth Average submenu.

*Log Frequency* 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. Frequency* sweep with a stimulus range between 50 MHz and 6 GHz, the forward transmission parameter S12 as measured quantity, and a *dB Mag* scaled y-axis




**Remote control:** `[SENSe<Ch>:]SWEep:TPYE LOGarithmic`
`[SENSe<Chn>:]FUNCtion[:ON] "XFRequency:..."`

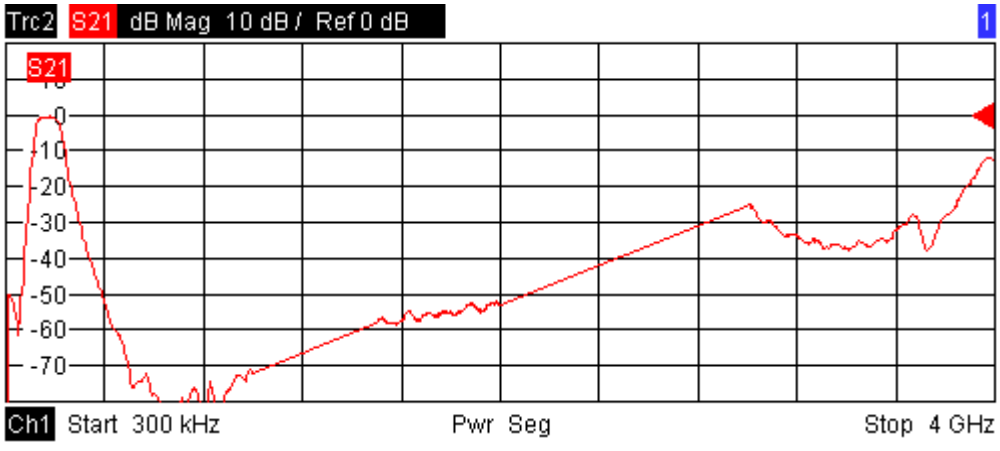
## Segmented Frequency

In a *Segmented Frequency* sweep the sweep range can be composed of several continuous, non-

overlapping frequency sub-ranges or single frequency points. The sub-ranges are termed sweep segments and defined in the Define Segments dialog. 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, the selectivity of the measurement filter, the frequency band of the local oscillator, and the measurement time can be set independently for the individual segments.

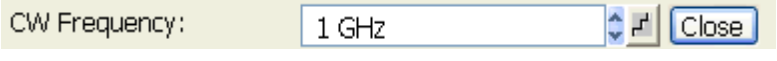
Due to this flexibility *Segmented Frequency* sweeps are suitable for any detailed analysis of a DUT at specified frequencies. In a Cartesian diagram the measurement result is displayed as a trace over a linear frequency scale ranging from the lowest to the highest frequency point of all segments. The following example shows a *Segmented Frequency* sweep with 3 segments in the stimulus range between 50 MHz and 6 GHz, the forward transmission parameter S12 as measured quantity, and a *dB Mag* scaled y-axis. In the frequency ranges between the sweep segments the trace is displayed as a straight line.




**Remote control:** `[SENSe<Ch>:]SWEep:TPYE SEGMent`
`[SENSe<Chn>:]FUNCtion[:ON] "XFRequency:..."`

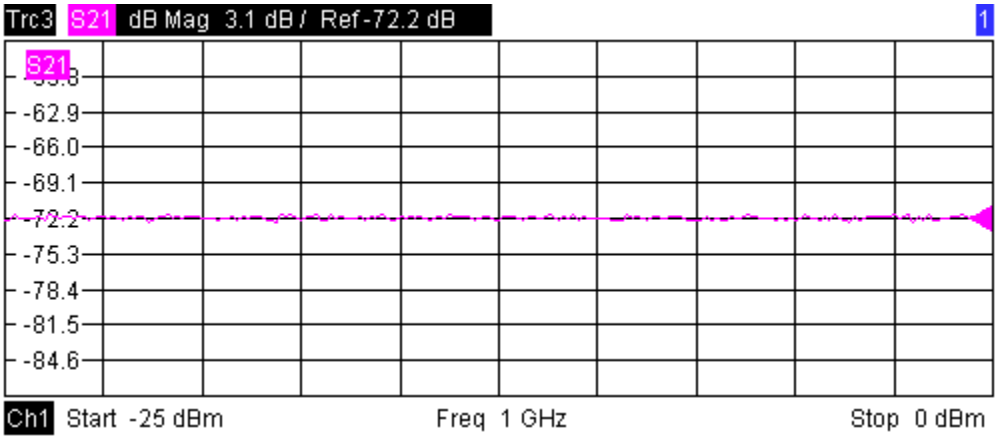
## Power

In a *Power* sweep the internal generator power is swept in linear, equidistant steps over a continuous power range. The generator power range (sweep range) is defined via *Channel - Stimulus - Start* and *Channel - Stimulus - Stop*; see Stimulus and Sweep Types. The numeric entry bar for the fixed frequency *(CW Frequency)* appears as soon as *Power* is activated.




*Power* sweeps are particularly suitable for the analysis of non-linear 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 with a linear DUT:




**Generator power**

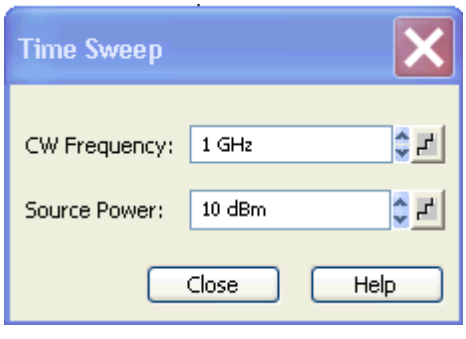
The power range defined for a power sweep replaces the internal generator power setting in the Power Bandwidth Averagesubmenu.

The power corresponds to the actual output power at the test port that supplies the stimulus for the active channel or at the calibrated reference plane. Any generator power calibration or attenuation of the active generator step attenuators is taken into account.The wide range of power stimulus values leaves enough flexibility to include an attenuation or gain in the test setup.

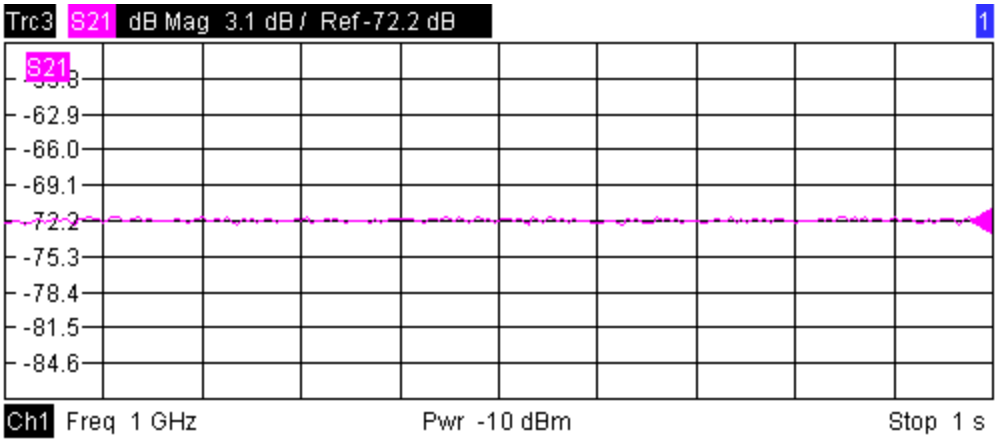
**Remote control:**
```
[SENSe<Ch>:]SWEep:TPYE POWer
SEGMent[SENSe<Chn>:]FUNCtion[:ON] "XPOWer:..."
SOURce<Ch>:FREQuency<Pt>:CW|FIXed
```

## Time

*Time* sweeps are performed at constant frequency and stimulus power. A single sweep extends over a specified period of time. The sweep time is defined via *Channel - Stimulus - Stop;* see Stimulus and Sweep Types. The time intervals between two consecutive sweep points are calculated according to <Stop>/(n - 1) where n is the selected *Number of Points.* The frequency *(CW Frequency)* and internal generator power *(Source Power)* is fixed and entered into a field which pops up as soon as *Time* is activated:




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 (oscillographic representation). The following example shows a *Time* sweep with a DUT that does not change its characteristics in time:



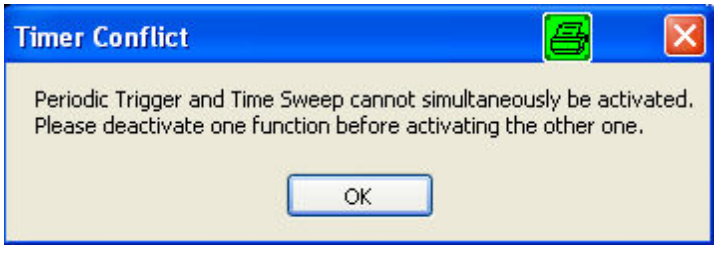

**Sweep time**

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, taking into account the current measurement settings.

If the total sweep time entered via *Channel - Stimulus - Stop* is smaller than the estimated minimum sweep time, the analyzer opens a dialog to adjust (increase) the stop time.

**Trigger settings**

The *Time* sweep is incompatible with the *Periodic Trigger*: If *Periodic...* is selected while a time sweep is active, the analyzer generates an error message:




To continue, click *OK* and select any other trigger source.

In *Time* sweep mode the *Start, Center* and *Stop* softkeys are inactive, still you can use the *START CENTER* hardkey to select the fixed CW frequency.

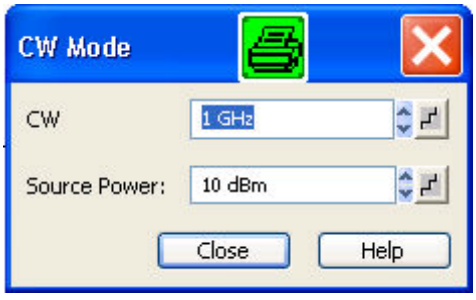
**Remote control:**
```
[SENSe<Ch>:]SWEep:TPYE CW
[SENSe<Chn>:]FUNCtion[:ON] "XTIMe:..."
SOURce<Ch>:FREQuency<Pt>:CW|FIXed
SOURce<Ch>:POWer[:LEVel][:IMMediate][:AMPlitude]
```

## CW Mode

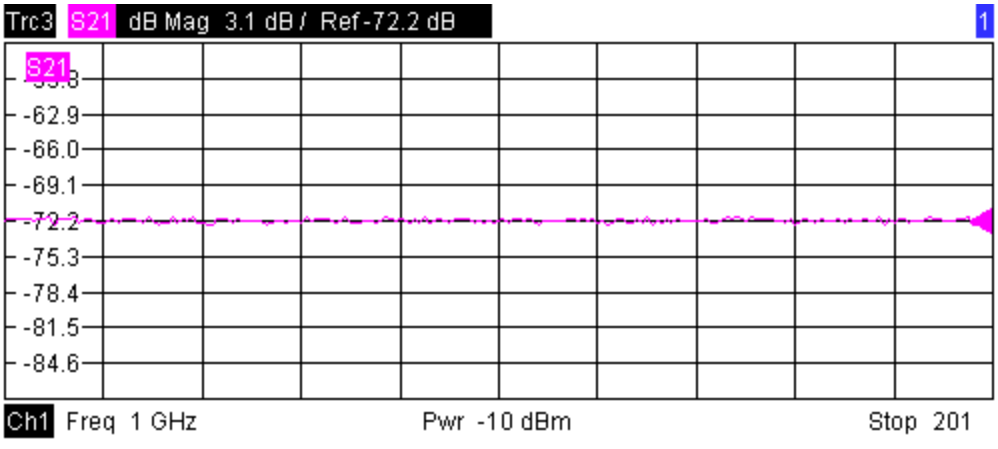
*CW Mode* sweeps, like *Time* sweeps, are performed at constant frequency and stimulus power. The measurement is triggered according to the current trigger settings; each trigger event triggers the first

partial measurement of a measurement point. The total number of measurement points is defined via *Channel - Stimulus - Stop*; see Stimulus and Sweep Types. The time interval between two consecutive measurements depends on the trigger settings. Any trigger mode is allowed.

The frequency *(CW)* and internal generator power *(Source Power)* is fixed and entered into a field which pops up as soon as *CW Mode* is activated.




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 (oscillographic representation). The diagram is analogous to the *Time* diagram. The following example shows a *CW Mode* sweep with a DUT that does not change its characteristics in time:




**Sweep time**

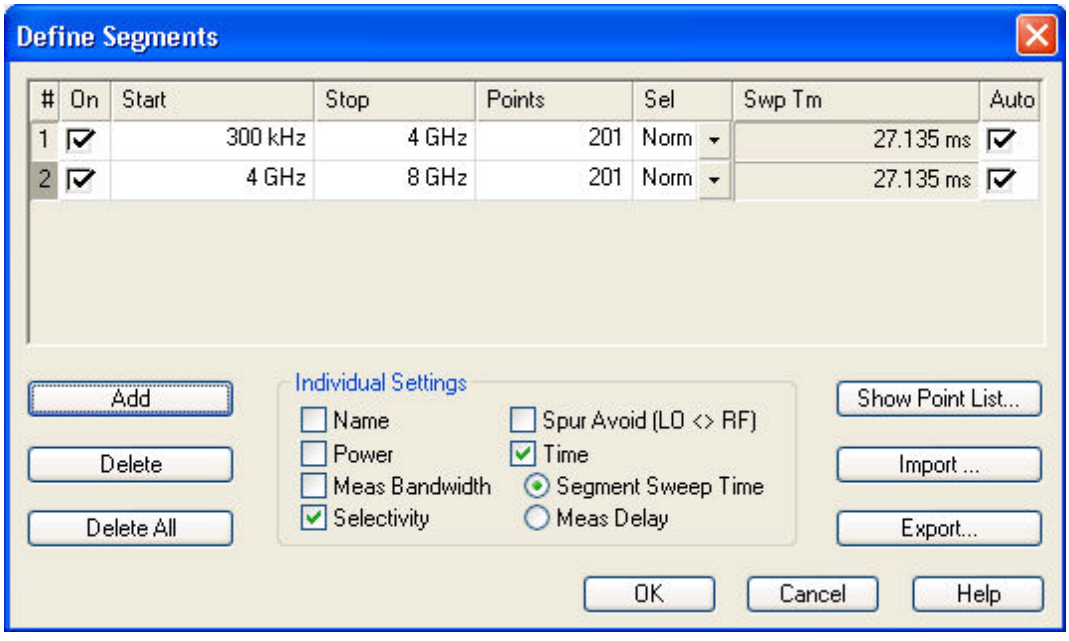
The time delay between two consecutive trigger pulses must not be smaller than the minimum measurement time for one measurement point described for Time sweeps.

In *CW Mode* sweep mode the *Start, Center* and *Stop* softkeys are inactive, still you can use the *START CENTER* hardkey to select the fixed CW frequency.

**Remote control:**
```
[SENSe<Ch>:]SWEep:TPYE POINt
[SENSe<Chn>:]FUNCtion[:ON] "XTIMe:..."
SOURce<Ch>:FREQuency<Pt>:CW|FIXed
SOURce<Ch>:POWer[:LEVel][:IMMediate][:AMPlitude]
```

## Define Segments

Opens a dialog to define all channel settings for a *Segmented Frequency* sweep and to import and export segmented sweep settings.




The *Define Segments* dialog contains a table to edit the individual segments of the sweep range. Sweep segments must not overlap, however, two adjacent segments can have one common point; see *Columns in the Define Limit Line table*.

Below the table, three groups of controls provide additional settings. Refer to the following sections below:

- Inserting and Deleting Segments
- Individual Segment Settings
- Point List, Segment Import and Export

**Columns** in the *Define Limit Line* table

The table contains an automatically assigned current number for each segment plus the following editable or non-editable columns:

| # | On | Start | Stop | Points |
|---|---|---|---|---|
| 1 | ☑ | 300 kHz | 19.999999996 GHz | 201 |
| 2 | ☑ | 19.999999996 GHz | 20 GHz | 201 |

- *On* provides check boxes to activate or deactivate each individual segment. Sweep points belonging to inactive segments only are not measured and not listed in the *Point List.*
- *Start* is the stimulus (x-axis) value of the first point of the segment. If the segment contains more than one *Point,* then *Start* must be smaller than the *Stop* value. If a *Start* value equal to or larger than the current *Stop* value is set, *Stop* is adjusted to the new *Start* value plus 1 Hz.
- *Stop* is the stimulus (x-axis) value of the last point of the segment. If the segment contains more than one *Point,* then *Stop* must be larger or equal than the *Start* value. If a *Stop* value equal to or smaller than the current *Start* value is set, *Start* is adjusted to the new *Stop* value minus 1 Hz.

- *Points* is the number of sweep points in the segment. A single segment can consist of only one point, however, the entire sweep range must contain at least 2 distinct frequency points. If *Points* is set to 1, then the *Stop* frequency is set equal to the *Start* frequency.
- The remaining columns show the channel settings for each segment. They are displayed only if they are selected in the *Individual Segment Settings* panel.

*When a new segment is defined or when the stimulus range of a segment is modified, the analyzer automatically checks whether it is compatible with the existing segments. The Start and Stop values of the existing segments are modified so that there is no overlap between any of the segments. If this is not possible (e.g. because of the limited frequency range of the analyzer), the entry of the new Start or Stop value is denied.*

Use the paste marker list for convenient entry of *Start* and *Stop* values.

**Remote control:** The commands in the `[SENSe<Chn>:]SEGMent...` subsystem define all sweep segment settings.

## Inserting and Deleting Segments

Three buttons on the left side below the table in the *Define Segments* dialog extend or shorten the segment list:

- **Add**

  adds a new segment to the list. The new segment is inserted after the active segment. The segment numbers (#) of all segments after the new segment are adapted.

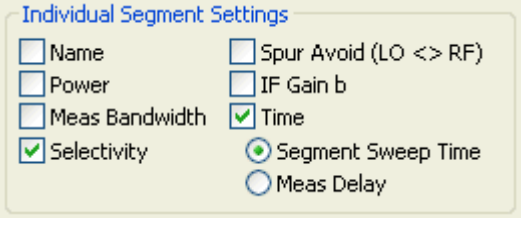
  To avoid overlapping segments, the new segment ranges from the *Stop* value of the previously active segment to either the *Start* valueof the next segment or the maximum frequency of the analyzer. The analyzer places no restriction on the number of segments in a sweep range.

- 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 control:**
```
[SENSe<Chn>:]SEGMent<Seg>:INSert
[SENSe<Chn>:]SEGMent<Seg>:DELete
[SENSe<Chn>:]SEGMent<Seg>:DELete:ALL
```

## Individual Segment Settings

The options in the *Individual Segment Settings* panel can be used to vary the channel settings of every individual segment in the list.




The first sweep segment is created with the channel settings defined for general sweep types. When any further sweep segment created, it uses the channel settings of the previously active segment. Each

selected (checked) option adds a column to the segment list.

- Name adds a column to assign a name to each segment. A segment name is a string that may contain letters, numbers and special characters.
- Power defines the internal source *Power* for each individual sweep segment.
- Meas Bandwidth defines the *Meas Bandwidth* for each individual sweep segment.
- Selectivity defines the *Selectivity* of the IF filter used for each sweep segment.
- Spur Avoid defines whether the analyzer measures the segment with a local oscillator frequency LO below or above the RF input frequency. The parameter replaces the *Spurious Avoidance* settings for a particular segment.
- **Time**

  defines the sweep time for each segment. The default configuration for a new segment is equal to the Sweep Time settings.

  When *Time* is checked two new columns appear in the table. The first column reads *Swp Tm* or *Pnt Dly,* depending on the selected radio button below the *Time* checkbox. The second column reads *A(uto)* and is used to activate automatic sweep time setting.

  - A(uto) minimizes the sweep time. If this option is checked, the columns *Swp Tm* or *Meas. Dly* can not be edited. *Swp Tm* indicates the estimated sweep time, taking into account the current measurement settings, the *Meas. Dly* is 0 s. The segment sweep time and point delay values are maintained until changed explicitly if *Auto* is switched off.
  - Swp Tm or Segment Sweep 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 the *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. Dalay* modifies the *Swp Tm* and vice versa.

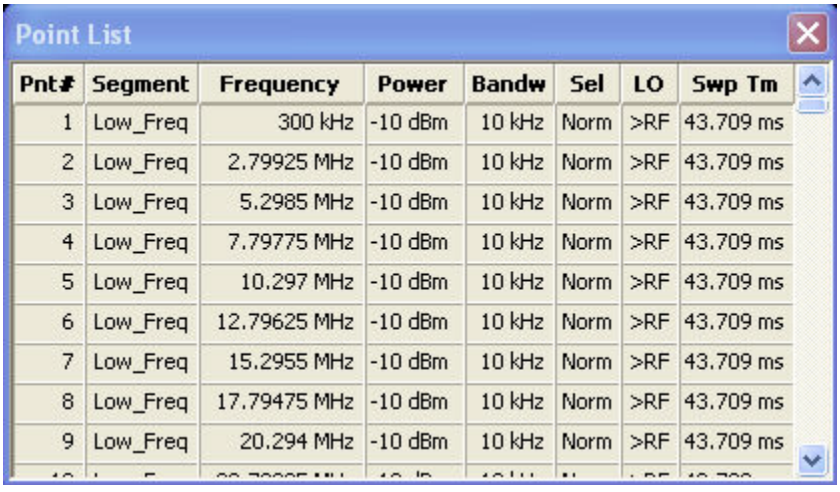
**Remote control:**
```
[SENSe<Chn>:]SEGMent<Seg>:POWer[:LEVel]
[SENSe<Chn>:]SEGMent<Seg>:BWIDth[RESolution]
[SENSe<Chn>:]SEGMent<Seg>:BWIDth[RESolution]:SELect
[SENSe<Chn>:]SEGMent<Seg>:DEFine
[SENSe<Chn>:]SEGMent<Seg>:INSert
[SENSe<Chn>:]SEGMent<Seg>:SWEep:TIME
[SENSe<Chn>:]SEGMent<Seg>:SWEep:DWELl
```

## Point List, Segment Import and Export

The buttons on the right side below the table in the *Define Segments* dialog are used to retrieve the position of the defined sweep points import and export limit line data.

- ***Show Point List***

  opens a list of all active sweep points and their channel settings.



Point List

| Pnt# | Segment | Frequency | Power | Bandw | Sel | LO | Swp Tm |
|---|---|---|---|---|---|---|---|
| 1 | Low_Freq | 300 kHz | -10 dBm | 10 kHz | Norm | >RF | 43.709 ms |
| 2 | Low_Freq | 2.79925 MHz | -10 dBm | 10 kHz | Norm | >RF | 43.709 ms |
| 3 | Low_Freq | 5.2985 MHz | -10 dBm | 10 kHz | Norm | >RF | 43.709 ms |
| 4 | Low_Freq | 7.79775 MHz | -10 dBm | 10 kHz | Norm | >RF | 43.709 ms |
| 5 | Low_Freq | 10.297 MHz | -10 dBm | 10 kHz | Norm | >RF | 43.709 ms |
| 6 | Low_Freq | 12.79625 MHz | -10 dBm | 10 kHz | Norm | >RF | 43.709 ms |
| 7 | Low_Freq | 15.2955 MHz | -10 dBm | 10 kHz | Norm | >RF | 43.709 ms |
| 8 | Low_Freq | 17.79475 MHz | -10 dBm | 10 kHz | Norm | >RF | 43.709 ms |
| 9 | Low_Freq | 20.294 MHz | -10 dBm | 10 kHz | Norm | >RF | 43.709 ms |

Points in inactive sweep segments (i.e. segments that are not switched *On* in the segment table) are not shown. The table provides a check of the settings made and can not be edited.

- *Import Segment List...* calls up an *Open File* dialog to load a sweep segment list from a sweep segment file. Sweep segment files are ASCII files with the default extension *.seglist and a special file format.
- *Export Segment List...* calls up a *Save As...* dialog to store the current sweep segments to a sweep segment file. Sweep segment files are ASCII files with the default extension *.seglist and a special file format.

To import a segment list file (*.seglist) you can also use the Windows Explorer and simply double-click the file or drag and drop the file into the NWA application. You must enable the segmented sweep separately.

**Remote control:** `MMEMory:LOAD:SEGMent <Ch_no>,"file_name"`
`MMEMory:STORe:SEGMent <Ch_no>,"file_name"`

## File Format for Sweep Segments

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 *Individual Segment Settings* that may be actually hidden.

**Example of a sweep segment file**

The segmented sweep range:

| # | On | Start | Stop | Points |
|---|---|---|---|---|
| 1 | ☑ | 300 kHz | 300.002 kHz | 401 |
| 2 | ☑ | 300.002 kHz | 300.004 kHz | 401 |
| 3 | ☑ | 4.5 GHz | 8 GHz | 401 |

is described by the following sweep segment file:

```
# Version 0
#
boState;        strName;        Start Frequency[MHz];   Stop Frequency[MHz];    intNo of Points;
true;           »;              0.300000;               2000.000000;401;        -10.000000;
true;           »;              2000.000000;            3000.000000;401;        -10.000000;
true;           »;              4500.000000;            8000.000000;401;        -10.000000;
```

The sweep segment file actually contains more columns listing all channel settings of the individual sweep segments. The headings of the additional columns read:

*Source Power [dBm]; IF Bandwidth [Hz]; enIF Selectivity; en IF Sideband; Meas Delay [µs]; boSweep Time Auto;*

## Pulsed Measurements (for R&S ZVA and R&S ZVT)

The softkeys in the lower part of the *Sweep Type* submenu control pulsed measurements (with option ZVA-K7, *Pulsed Measurements)*.

**Pulse profile mode**

Measurements on pulsed RF signals are required in many areas of RF and microwave technology. Pulsed signals are used in mobile phone applications and radar systems, and amplifiers are typically designed for pulsed rather than continuous wave (CW) conditions.

The analyzer performs pulsed measurements in analogy to a time sweep (i.e. at constant receiver frequency), but with a much higher sampling rate of 12.5/ns. The raw I/Q amplitudes are written into a ring buffer and processed at the end of each sweep. The buffer size allows for a maximum recording time (sweep time) of 3 ms. Due to the high sampling rate and the large IF bandwidths available, it is possible to obtain profiles for pulse widths from approx. 200 ns to the maximum recording time. Of course it is also possible to measure a sequence of pulses up to the maximum recording time.

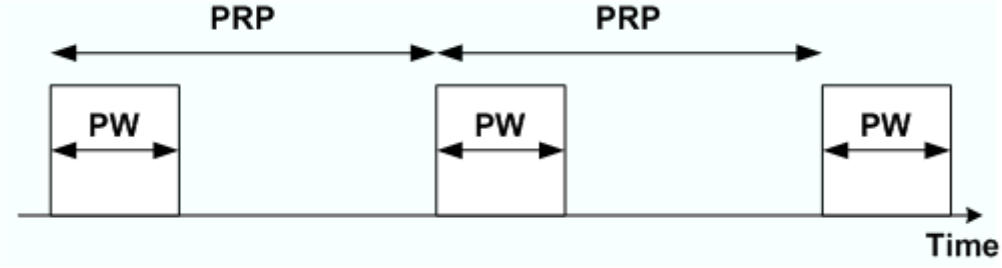
Pulsed measurements require a trigger signal that is synchronized to the analyzed pulses. The analyzer's trigger system is idle while the data acquired in the last recording period is processed; it is re-armed after data processing is finished. To suppress the noise occurring at high IF bandwidths, sweep averaging is recommended.

The RF source signal of the vector network analyzer is continuous, so the pulsed stimulus signals must be generated by external means:

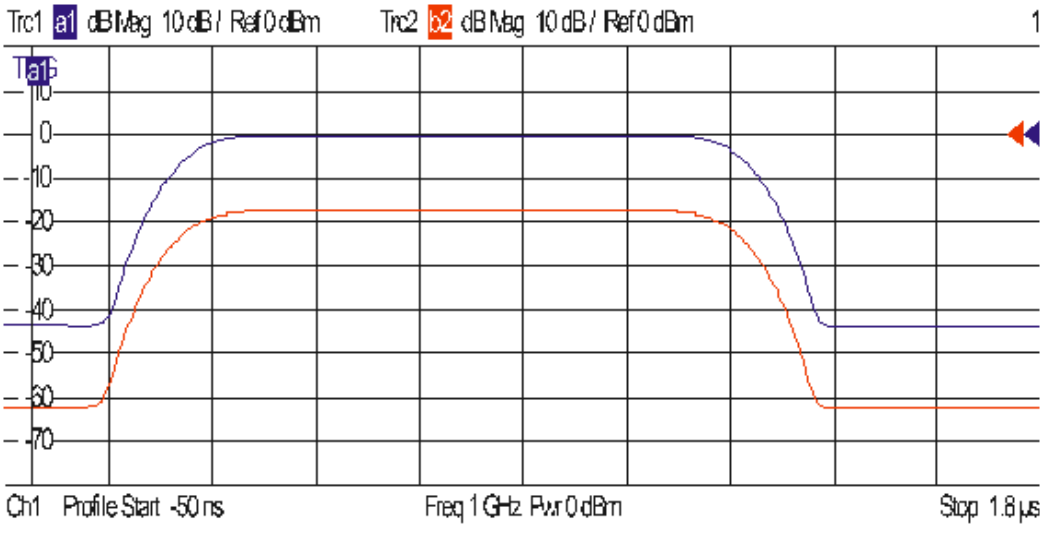
- A chopper between the SOURCE OUT and the SOURCE IN connector of an analyzer port transforms the continuous source signal into a pulsed signal.
- An external generator can provide pulsed signals.
- The DUT can be switched on and off periodically to show a pulsed characteristics.

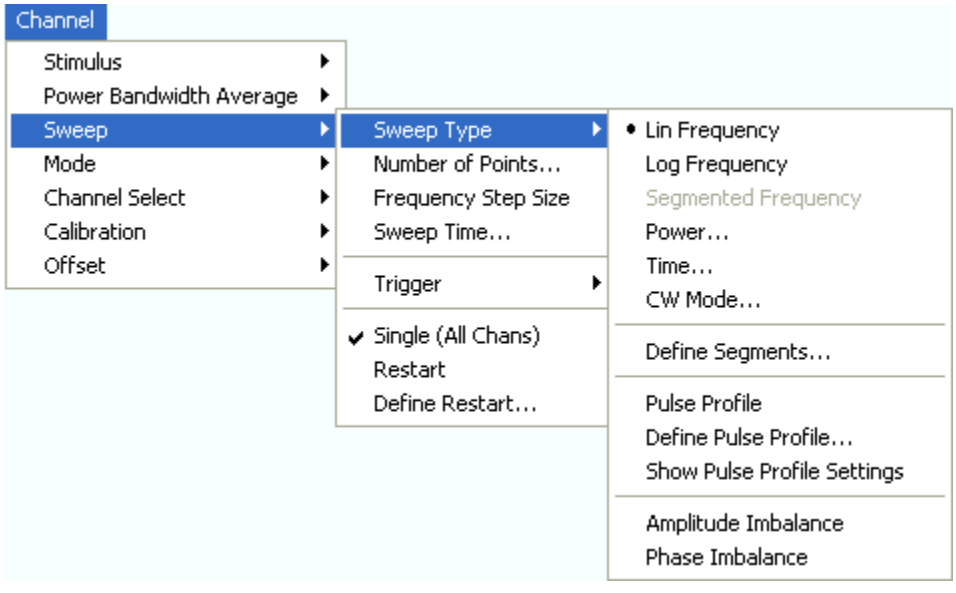
**Basic relations: pulse width, repetition interval, IF bandwidth**

A pulsed RF signal ideally consists of a sequence of periodic rectangular pulses. The pulse width PW and pulse repetition period PRP are shown in the following figure; the ratio PW/PRP is often referred to as the duty cycle.




The spectrum of a pulsed signal in the frequency domain has a sin(x)/x envelope, where x is proportional to the duty cycle. The spectral components are equidistant with a spacing of 1/PRP. The analyzer measures and displays the pulses in time domain. To be detected properly, the pulse width must be larger than twice the rise/fall time of the receiver, which roughly corresponds to the inverse of the IF bandwidth. The figure below shows a rectangular pulse with a width of 1 µs, measured in a 10 MHz bandwidth. The rise and fall times are approximately 100 ns.



- *Pulse Profile* activates pulsed measurements.
- *Define Pulse Profile...* gives access to the time parameters, the stimulus signal settings, and the receiver settings for pulsed measurements.
- *Show Pulse Profile Settings* opens a screen with an overview of the current settings.

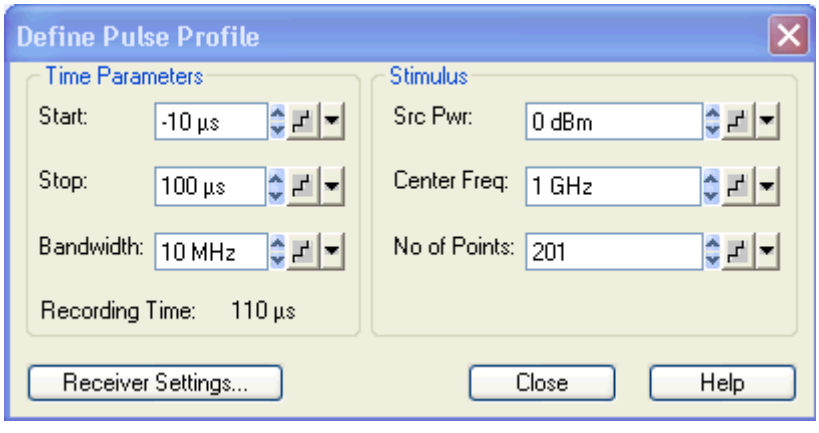
## Pulse Profile

*Pulse Profile* measurements are performed at constant receiver frequency and stimulus power; see background information on pulse profile mode. Pulsed measurements require a trigger signal that is synchronized to the analyzed pulses. Besides all settings are accessible in the *Define Pulse Profile* dialog.

**Remote control:** `[SENSe<Ch>:]SWEep:TPYE PULSe`

## Define Pulse Profile...

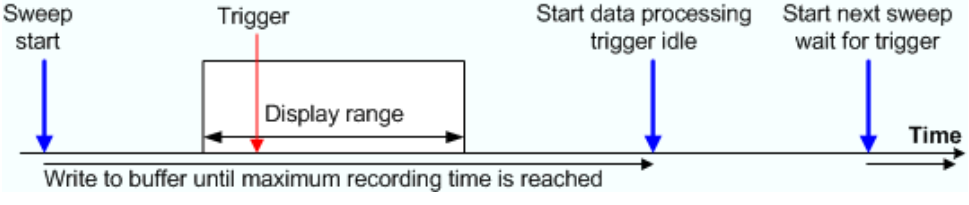
Opens a dialog to configure the time parameters, the stimulus signal settings, and the receiver settings for pulsed measurements.




The *Time Parameters* define the displayed time range and the IF bandwidth:

- *Start* and *Stop* define the start and stop time in the diagram area relative to the trigger time; they are identical to the *Stimulus – Start* and *Stimulus – Stop* parameters. The stop time must be larger than the start time; **negative values** are allowed.

  When the pulse profile measurement is activated (and then every time when the data of the last sweep has been processed) the analyzer immediately starts acquiring data, therefore it is possible to select negative start times (pre-trigger). Moreover, the time of the trigger event does not have to be within the displayed time range (i.e. start and stop time can both be negative or positive).




- *Bandwidth* selects the IF bandwidth for the pulse profile measurements. The measurements are performed with a fixed sampling rate and at fixed frequency, so the bandwidth does not affect

the measurement speed. To obtain short rise and fall times of the receiver and measure short pulses, it is preferable to use large bandwidths; see background information on basic relations.

*Filter bandwidths above 10 MHz can be associated with increased measurement uncertainties. In particular, they tend to cause overshoot and ringing at the beginning and at the end of the pulses.*

- *Recording Time* indicates the width of the display range, i.e. the difference of *Stop – Start* time. The calculated recording time is possibly modified if *Shift Stimulus* or *Coupled Section Limit Lines On* is active; see *Receiver Settings.* An error message is displayed if the calculated recording time exceeds the maximum buffer recording time of 3 ms.

The *Stimulus* parameters define fixed source and receiver settings for the pulse profile measurement. The parameter settings are also valid for other sweep types.

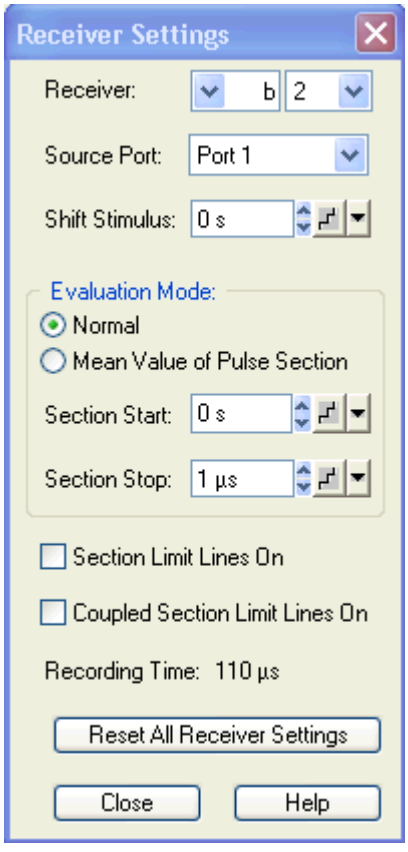
- *Source Power* is the channel base power; see *Power*. This setting is relevant if the analyzer is used to generate the pulsed source signal, e.g. with an additional RF switch looped into the SOURCE path.
- *Center Freq.* is the channel base frequency; see *CW Frequency*. Note that pulse profile measurements are compatible with the frequency-converting mode, so that it is possible to define port-specific corrections to the CW frequency.
- *No of Points* is the number of sweep points displayed; see *Number of Points*. Due to the fixed sampling rate of 12.5 ns and the fixed buffer size, this parameter does not directly affect the measurement speed. Results are interpolated if the number of points differs from the value (Stop – Start)/12.5 ns + 1.

*Receiver Settings* opens a dialog with advanced pulse profile settings.

**Remote control:**
```
[SENSe<Ch>:]PULSe:TIME...
SOURce<Ch>:POWer<Pt>[:LEVel][:IMMediate][:AMPlitude]
SOURce<Ch>:FREQuency<Pt>[:CW]|FIXed
[SENSe<Ch>:]SWEep:POINts
```

## Receiver Settings

This dialog provides advanced settings for the pulse profile mode. Most of the parameters can be set independently for the different receivers and source ports involved in the measurement.




The receiver and source port are selected in the upper part of the dialog.

- *Receiver* selects a receiver for a received wave $b_i$ or a reference wave $a_i$.
- *Source Port* contains all analyzer ports and external generators that are available as source ports.

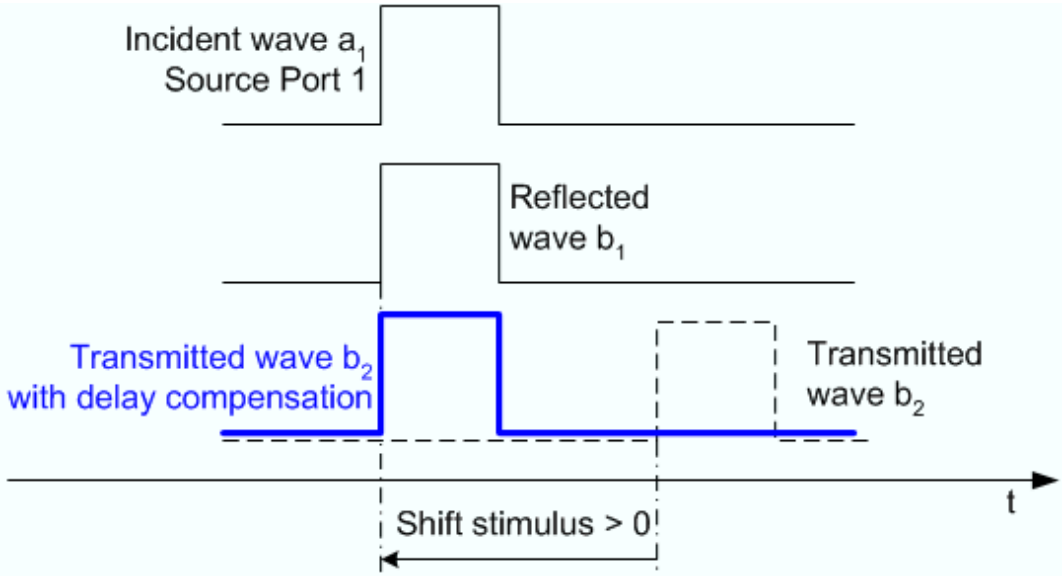
Receiver and source ports scan be selected although they may not be currently used. The analyzer can store predefined receiver settings for arbitrary port configurations.

- *Shift Stimulus* defines an offset time for the current receiver and source port. This is important for measuring DUTs with a noticeable transmission delay.

**Effect of shift stimulus and examples**

*Shift Stimulus* defines an offset time that is to compensate for a known transmission delay of a DUT. A non-zero shift stimulus value has the following effect:

- The trace for the wave quantity assigned to the active receiver is shifted in horizontal direction. A positive value shifts the trace to the left (subtraction of a known delay), a negative value shifts it to the right.
- All quantities depending on the shifted wave quantity are calculated with the corrected time reference. In the example below, the S-parameter $S_{21}$ is calculated as $S_{21}(t) = b_2(t + <\text{Shift stimulus}>)/a_1(t)$.
- The displayed *Recording Time* is extended to cover the delayed wave.




The example above illustrates the importance of a correct delay compensation for the calculation of S-parameters, ratios and other derived quantities in the time domain. Without compensation, the two pulses for $a_1$ and $b_2$ do not overlap so that no meaningful S-parameter calculation is possible.

**Example:** S-parameter measurement for a DUT with known delay $D_{fw}$ and $D_{rev}$ in forward and reverse direction. The DUT is connected between port 1 and port 2 of the analyzer. The test setup is fully calibrated so that the reference planes are at the DUT's input and output ports.

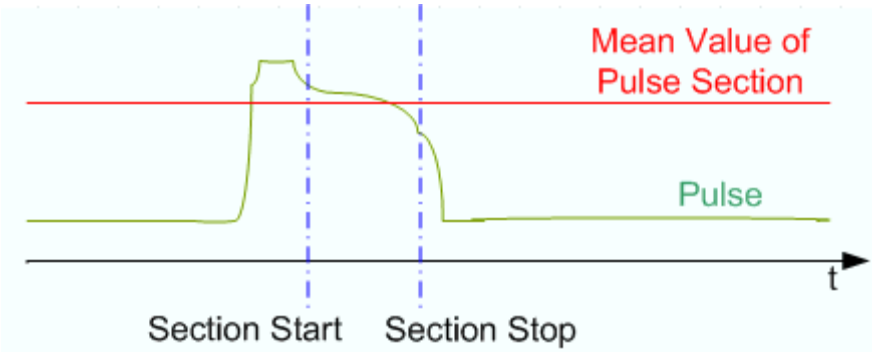
| | | |
|---|---|---|
| Receiver: a1<br>Source: 1<br>Shift stimulus: 0 | Receiver: b1<br>Source: 1<br>Shift stimulus: 0 | Receiver: b2<br>Source: 1<br>Shift stimulus: $D_{fw}$ |
| Receiver: a2<br>Source: 2<br>Shift stimulus: 0 | Receiver: b1<br>Source: 2<br>Shift stimulus: $D_{rev}$ | Receiver: b2<br>Source: 2<br>Shift stimulus: 0 |

- The *Evaluation Mode* specifies how the wave quantity assigned to the current receiver and source port is displayed. In *Normal* mode, it is displayed as measured (pulse shape). If *Mean Value of Pulse Section* is selected, the wave quantity is replaced by its mean value (magnitude and phase) averaged over a configurable time interval [Section Start, Section Stop].

**Effect of averaging, definition of the time interval**

*Mean Value of Pulse Section* has the following effect:

- The trace for the wave quantity assigned to the active receiver is replaced by a constant trace.
- All quantities depending on the wave quantity are based on the averaged values.
- The displayed *Recording Time* is extended to cover the specified pulse section.




*Section Start and Section Stop can be negative and even outside the displayed time range, however, Section Start must be smaller than Section Stop. A stimulus shift also shifts the Section Start and Section Stop values.*

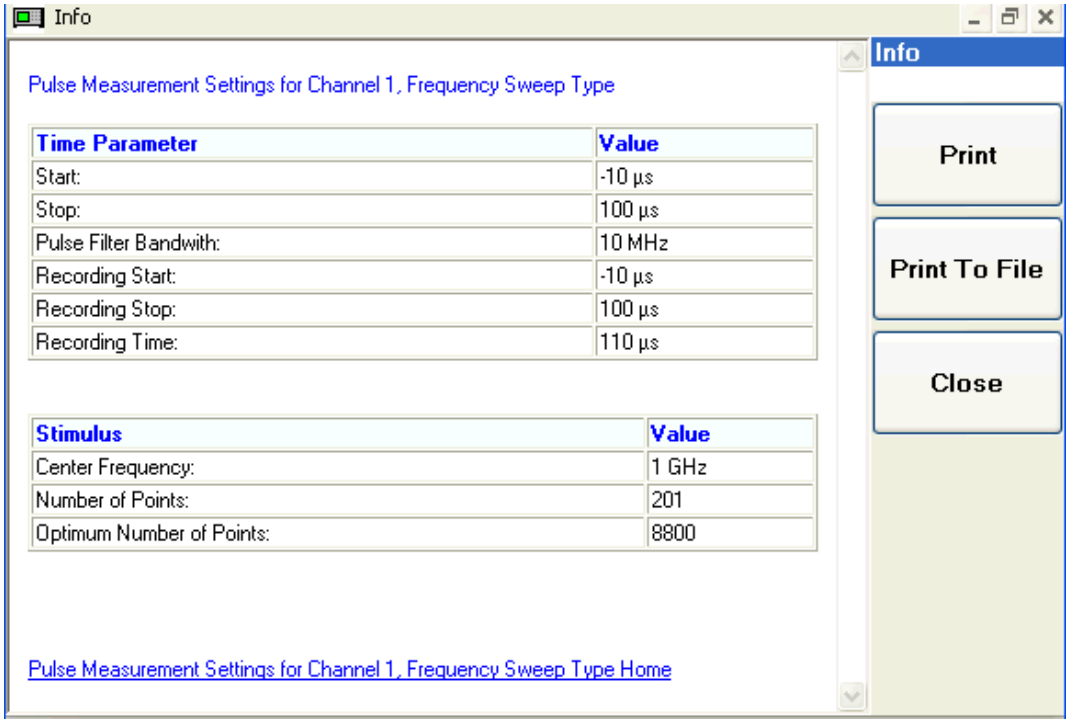
The remaining settings in the dialog are general; they do not depend on the selected receiver and source port.

- *Section Limit Lines On* places two vertical lines into the diagram(s), indicating the selected *Section Start* and *Section Stop*. The section limits can be modified using drag-and-drop.
- *Coupled Section Limit Lines On* couples the section limits (and the section limit lines) for all receivers and source ports and in all channels. The evaluation mode and the *Shift Stimulus* values are still independent. As a typical application, it is possible to create two channels, display the same wave quantity with different evaluation modes (pulse shape and average), vary the section limits in normal evaluation mode (pulse shape) and monitor the effect on the average trace.
- *Recording Time* indicates the width of the display range, possibly modified if *Shift Stimulus* or *Coupled Section Limit Lines On* is active; see background information above. An error message is displayed if the calculated recording time exceeds the maximum buffer recording time of 3 ms.
- *Reset All Receiver Settings* the parameters in the dialog for all receivers and source ports.

**Remote control:** `[SENSe<Ch>:]PULSe:COUPled[:STATe]`
`[SENSe<Ch>:]PULSe:RECeiver...`

## Show Pulse Profile Settings

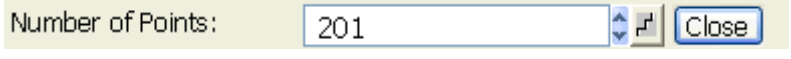
Gives an overview of the current time and stimulus settings for pulsed measurements. The info screen also display softkeys to *Print* the contents, save them to a file *(Print to File)* or *Close* the screen.




**Remote control:** Settings can be retrieved using the query form of the commands.

## Number of Points

Sets the total number of measurement points per sweep. The minimum number of points is 2.




Together with the sweep range defined with the *Stimulus* settings, this parameter defines the grid of sweep points. The sweep points are equidistantly distributed over the entire sweep range: The step width between two consecutive sweep points is constant on a linear scale (sweep types *Lin. Frequency, Time* and *CW Mode)* or on a logarithmic scale (sweep types *Log. Frequency* and *Power)*.

In *Segmented Frequency* sweeps, the number of points can be set independently for each segment; see *Define Segments*.

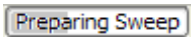
As an alternative to the *Number of Points,* the *Stimulus Step Size* can be set.

**Measurement time and screen resolution**

A large number of points improves the resolution of the trace but increases the measurement time.

The overall measurement time is composed of a hardware settling time at the beginning of the sweep plus the sum of the measurement times at each individual sweep point. This implies that the measurement time increases roughly linearly with the number of points.

*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 visualized by a Preparing Sweep symbol 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.*

**Remote control:** `[SENSe<Ch>:]SWEep:POINts`

## Frequency Step Size

Sets the distance between two consecutive frequency sweep points.

The step size is an alternative to the *Number of Points* setting:

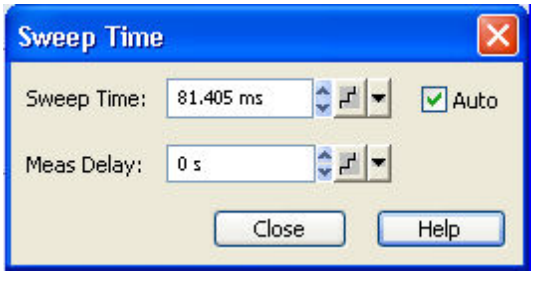
- If the sweep range is defined by means of the *Start* and *Stop* variables, both the *Stop* value and the *Number of Points* can vary as the *Stimulus Step Size* is changed. The *Stop* value is changed as little as possible so that the condition *Stimulus Step Size = (Stop – Start) / (Number of Points – 1)* can be fulfilled. Changing the *Start and Stop* values modifies the *Stimulus Step Size.*
- If the sweep range is defined by means of the *Center* and *Span* variables, both the *Span* value and the *Number of Points* can vary as the *Stimulus Step Size* is changed. The *Span* is reduced as little as possible so that the condition *Stimulus Step Size = (Stop – Start) / (Number of Points – 1)* can be fulfilled. Changing the *Span* modifies the *Stimulus Step Size.*

*This setting is valid for linear frequency sweeps only. It does not apply to logarithmic and segmented sweeps, power, time or CW Mode sweeps. Increasing the Frequency Step Size generally increases the measurement time.*

**Remote control:** `[SENSe<Ch>:]SWEep:STEP`

## Sweep Time

Opens a dialog to vary the measurement time for a sweep or delay the start of each sweep.




- *Auto* minimizes the sweep time. If this option is checked, the input fields *Sweep Time* and *Meas. Delay* can not be edited. *Sweep Time* indicates the estimated sweep time, taking into account the current measurement settings, the *Meas. Delay* is 0 s. The *Sweep Time* and *Meas. Delay* values are maintained until changed explicitly if *Auto* is switched off.
- *Sweep Time* is the total measurement time for the sweep. The minimum sweep time to be set is equal to the estimated value in the *Auto* mode. Setting a larger sweep time is equivalent to defining a *Meas. Delay* before each partial measurement.
- *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.

**Meas. Delay**

A delay time before the start of each partial measurement increases the accuracy, in particular in measurements on DUTs with long settling times (e.g. quartzes, SAW filters). As an alternative to increasing the delay (and thus the total sweep time), it is possible to select *Alternate* sweep.

The *Auto* and *Sweep Time* settings are not active in the *Time* and *CW Mode* sweep types. The *Meas. Delay* is taken into account for all sweep types.

In *Segmented Frequency* sweeps, the sweep time parameters can be set independently for each segment; see *Define Segments*. *Sweep Time* opens a message box indicating to total sweep time, i.e. the sum of the individual sweep segment times.

*In general, the system error correction is no longer valid after a change of the sweep time. The message Cal? appears in the trace list.*

**Remote control:** [SENSe<Ch>:]SWEep:TIME:AUTO
[SENSe<Ch>:]SWEep:TIME
[SENSe<Ch>:]SWEep:DWELl

## Trigger

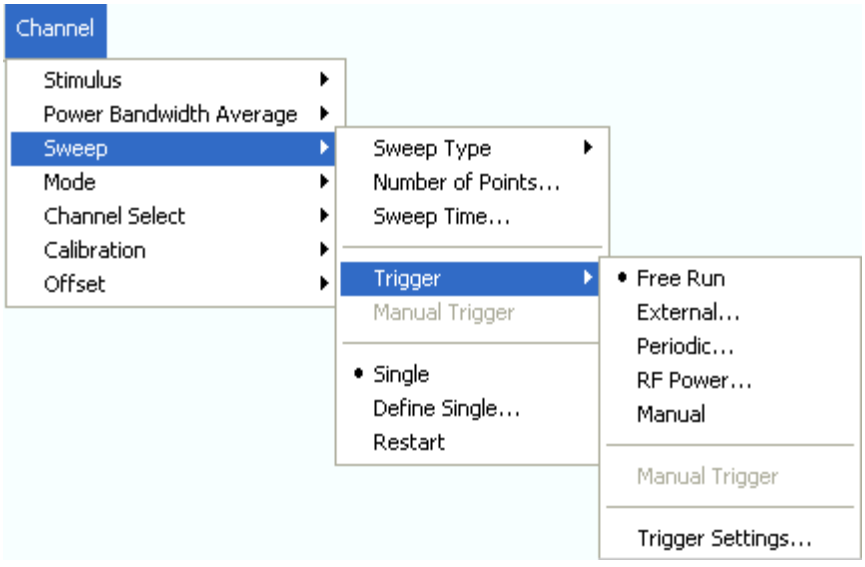
The *Trigger* submenu selects the source of the trigger signal and provides additional *Trigger Settings*.

**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 Trigger*). Triggered measurements are an alternative to the default mode *(Free Run, Continuous Sweep)*, where the measurement is continuously repeated without fixed time reference.

Any trigger event may 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.

*The trigger settings are also valid for calibration sweeps (see Measure Standards dialog). This means that, 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 Free Run trigger type.*




- *Free Run* is the default trigger mode. A new measurement is started immediately without waiting for a trigger signal and without fixed time reference.
- The *External* trigger signal is applied to the *EXT TRIGGER* or *USER CONTROL* connector at the rear panel.
- The *Periodic* trigger signal is provided by an internal clock generator.
- In *Manual* trigger mode, the trigger signal is generated on pressing the *Manual Trigger* softkey.
- *Trigger Settings* opens a dialog to specify the operation that the analyzer carries out after receiving a trigger event.

## Free Run

In *Free Run* mode a new measurement is started immediately without waiting for a trigger event and without fixed time reference. The *Trigger Settings* are not valid.

*Free Run* means that a measurement in *Continuous Sweep* mode is repeated as fast as possible.

**Remote control:** `TRIGger[:SEQuence]:SOURce IMMediate`

## External

In *External* trigger mode the measurement is triggered by a low-voltage (3.3 V) external TTL signal applied either to the BNC connector *EXT TRIGGER* or to pin 2 of the *USER CONTROL* connector at the rear panel. The two trigger inputs are equivalent; no additional setting for signal routing is required.

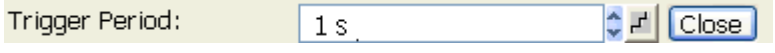
Selecting *External* opens a dialog to select whether the rising or the falling edge of the external trigger signal provides the trigger event.

*The period of the external trigger signal should be adjusted to the triggered measurement sequence. If the analyzer receives a trigger event while the last sequence is still running, the trigger event is skipped and a message Trigger event missed is displayed.*

**Remote control:** `TRIGger[:SEQuence]:SOURce EXTernal`
`TRIGger[:SEQuence]:SLOPe`

## Periodic

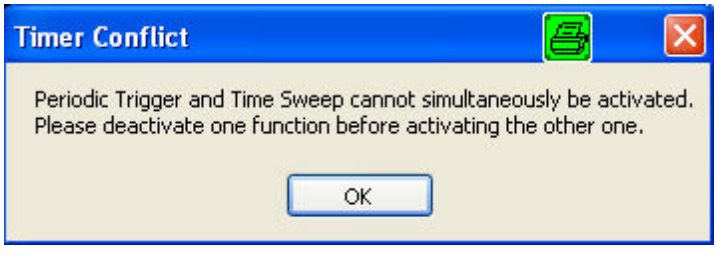
In *Periodic* trigger mode the measurement is triggered by the periodic signal of an internal clock generator. Selecting *Periodic* opens the numeric entry bar to select the trigger period.




The minimum trigger period that can be set corresponds to the estimated duration of the triggered measurement sequence.

**Time sweep**

The periodic trigger is incompatible with *Time* sweeps: If a *Time* sweep is selected while the periodic trigger is active, the analyzer generates an error message:




To continue, click *OK* and select any other trigger or sweep type (e.g. *CW Mode*).

**Remote control:** `TRIGger[:SEQuence]:SOURce TIMer`
`TRIGger[:SEQuence]:TIMer`

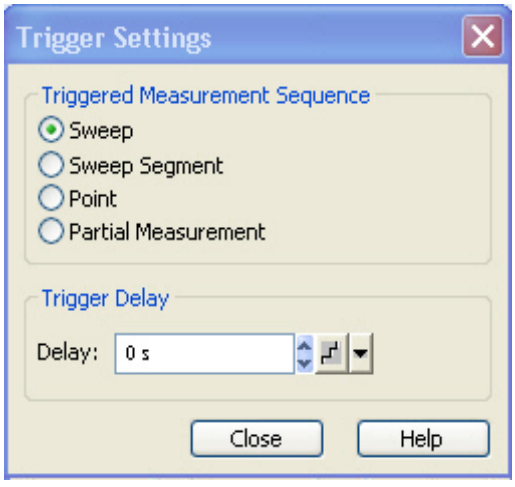
## Manual

In *Manual* trigger mode the trigger signal is generated on pressing the *Manual Trigger* softkey. *Manual Trigger* is disabled unless manual trigger mode is active.

**Remote control:** `TRIGger[:SEQuence]:SOURce MANual`
`*TRG`

## Trigger Settings

*Trigger Settings* opens a dialog to specify the operation that the analyzer carries out after receiving a trigger event.



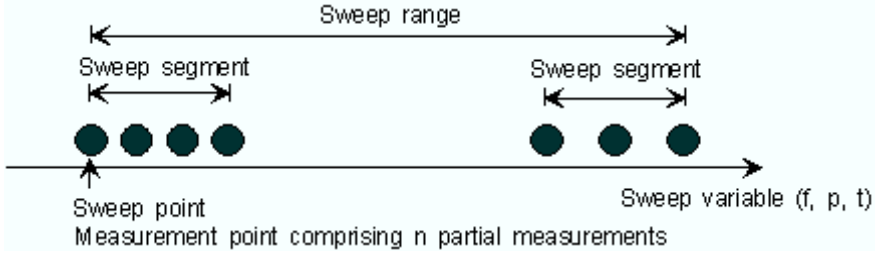

The dialog is divided into the following panels (for a detailed description refer to the following sections):

- *Triggered Meas. Sequence* selects the measurement cycle or sequence of actions to be triggered.
- *Trigger Delay* specifies a delay time between the trigger event and the start of the measurement.

*The Trigger Settings are valid for all trigger modes except Free Run. Some trigger settings are incompatible with other measurement settings; see below.*

## Triggered Meas. Sequence

Selects the measurement cycle or sequence of actions to be triggered.




- *Sweep* means that each trigger event starts an entire sweep, according to the current sweep configuration.
- *Sweep Segment* means that each trigger event starts the next sweep segment within the current sweep. If a sweep type other than *Segmented Sweep* is active, this option is equivalent to *Sweep*.
- *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 each measurement points requires only one partial measurement , this option is equivalent to *Point.*

**Relation to other sweep settings**

Some sweep settings are logically incompatible with a particular selection of the *Triggered Meas. Sequence:*

- If a *Time* sweep is performed, the *Triggered Meas. Sequence* is always a sweep.
- *Alternate* sweep mode only makes sense if the triggered measurement sequence comprises more than one sweep point. If *Point* or *Partial Measurement* is selected, *Alternate* sweep mode is switched off and vice versa.

*The period of the trigger events should be adjusted to the triggered measurement sequence. If the analyzer receives a trigger event while the last sequence is still running, the trigger event is skipped and a message Trigger event missed is displayed.*

**Remote control:** `TRIGger[:SEQuence]:LINK`

### Trigger Delay

Specifies a delay time between the trigger event and the start of the measurement.

The Delay time entered must be zero or positive, indicating that the trigger event precedes the start of the measurement (Posttrigger).

**Remote control:** `TRIGger[:SEQuence]:HOLDoff`

## Single (All Chans)

Toggles between single sweep and continuous sweep mode.

- In continuous mode *(Single (All Chans)* not selected), the analyzer measures continuously, repeating the current sweep.
- In single sweep mode, the measurement is stopped after the number of sweeps selected in the *Define Restart* dialog.

Press *Single* to activate the single sweep mode and start a single sweep sequence. To start further sweep sequences press *Restart.*

**Remote control:** `INITiate<Ch>:CONTinuous ON | OFF`

## Restart

Stops the current measurement and restarts a measurement sequence. In *Single* sweep mode a new single sweep sequence is started.

**Remote control:**

`INITiate<Ch>[:IMMediate]`

### Define Restart

Opens a dialog to configure the *Single* sweep mode.

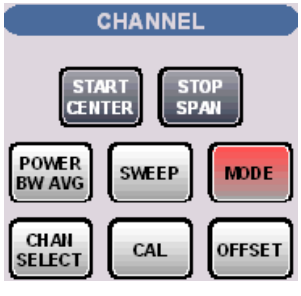
- *Number of Sweeps* selects the number of sweeps to be measured in single sweep mode: either one *(Single Sweep)* or a group of consecutive sweeps.
- *Scope* defines whether the active sweep mode (single or continuous) and the *Number of Sweeps* are valid for the *Active Channel* only or for *All Channels* in the active setup. If *All Channels* is selected, the number of sweeps in the single sweep sequence is equal to the selected *Number of Sweeps* times the number of channels. The sequence starts with the first sweep in channel no. 1.

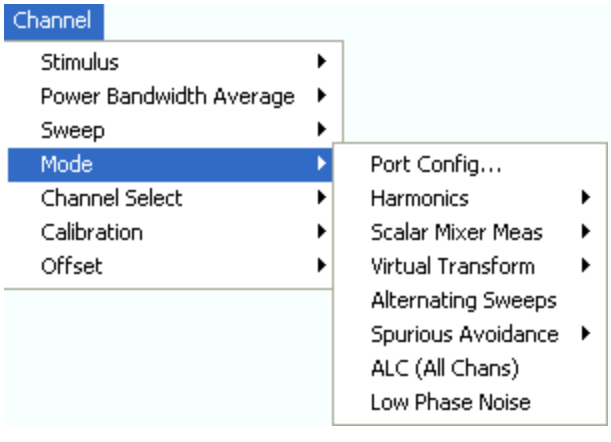
In remote control, it is possible to retrieve the results acquired in any of the sweeps within a single sweep group; see *Sweep History.*

**Remote control:** `[SENSe<Ch>:]SWEep:COUNt`
`INITiate<Ch>:IMMediate:SCOPe`

## Mode

The *Mode* menu defines the properties of the physical and logical (balanced) test ports, configures the source signals and the receiver, and provides special measurement modes.



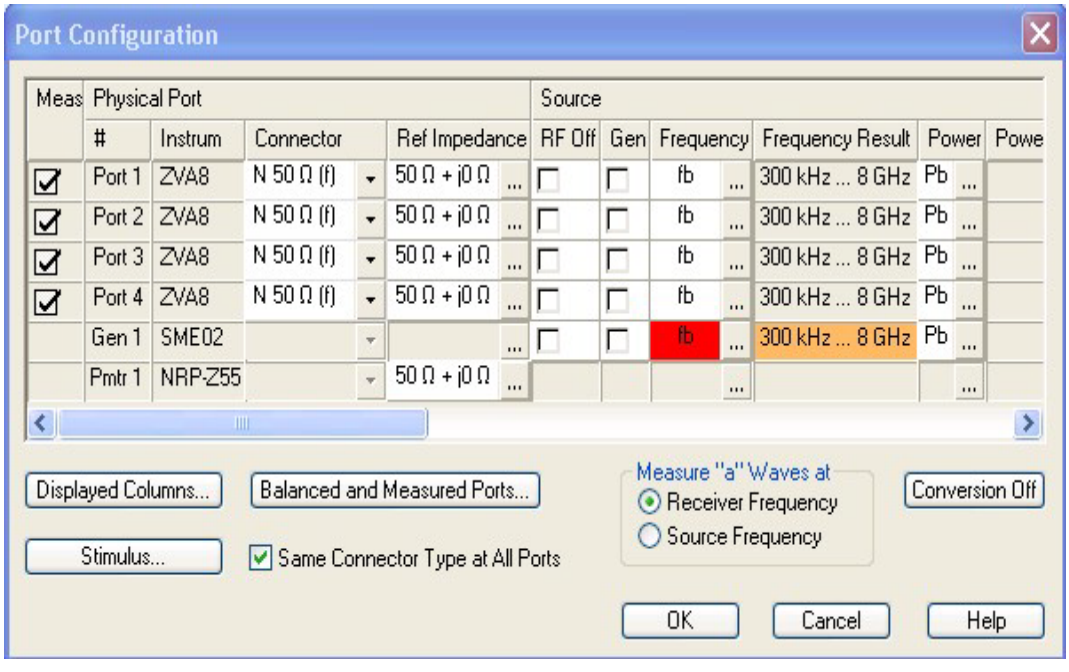




- *Port Config...* calls up a dialog to define the properties of the test ports, the source and the receiver.

- *Harmonics* provides arbitrary harmonic measurements.
- *Scalar Mixer Meas* measures the mixing products generated by an external mixer that is supplied with two independent RF signals.
- *Virtual Transform* defines the parameters of a virtual matching network to be embedded into/deembedded from the measurement circuit for a DUT with single ended or balanced ports.
- *Alternating Sweeps* activates or deactivates the operating mode where the analyzer performs only one partial measurement per sweep.
- *Spurious Avoidance* defines the relative position of the local oscillator (LO) relative to the RF input signal.
- *ALC (All Chans)* enables or disables the Automatic Level Control for all channels.
- *Low Phase Noise* reduces the phase noise of the source signals.

## Port Configuration

The *Port Configuration* dialog defines the properties of the test ports and configures the source signals and the receiver.




The table in the *Port Configuration* dialog contains the individual test port properties. Each table row corresponds to one logical or physical port; the number of rows is equal to the number of test ports of the analyzer.

Many measurement tasks can be performed with default port settings. Consequently the *Port Configuration* dialog does not show all columns in its default configuration. Use the *Displayed Columns* button to customize the table and display or hide columns.

**Measured Ports *(Meas/Group)***

The ports that can be used for the measurement (as selected in the *Balanced and Measured Ports* dialog) are indicated in the first column of the *Port Configuration* table *(Meas)*. The selected ports are used for the calculation of derived (mixed mode, Z, Y) parameters; they can be source ports and/or receive ports.

A measured port can be configured as a pure receive port by selecting *RF Off* in the *Source* section.

If one or more port groups to be measured are defined in the *Define Measured Ports* tab of the *Balanced and Measured Ports* dialog, then the column header changes from *Meas.* to *Group* and the column shows the group names (capital letters).

The settings are grouped together in four sections:

- *Logical Port* shows the properties of the balanced ports after they have been defined in the *Balanced and Measured Ports* dialog. This section is hidden if no balanced ports are defined.
- *Physical Port* shows and/or defines the properties of the physical (unbalanced) ports of the analyzer.
- *Source* shows and/or defines the generator settings at each physical port of the analyzer.
- *Receiver* shows and/or defines the receiver settings at each physical port of the analyzer.

Below the table, three buttons open additional dialogs:

- *Displayed Columns...* customizes the port configuration table by displaying or hiding columns.
- *Balanced and Measured Ports...* selects a balanced port configuration and defines the reference impedances at the ports.
- *Stimulus* modifies all sweep frequencies and powers in the active channel, based on an arbitrary frequency and power reference.

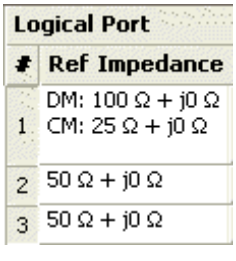
The dialog contains the following additional controls:

- If *Same Connector Type at All Ports* is active, the connector types at all ports (but not their gender) are always adjusted to the current selection in the *Connector* column.
- *Measure "a" Waves at* selects the analyzer frequency for the measurement of the reference waves $a_n$. Depending on the measured quantity, the reference waves can be measured at the receiver or at the source frequency.
- *Conversion Off* disables all power and frequency-converting modes, i.e. it resets all port frequencies to the basic channel frequency, sets the power to the base power of the channel, and disables the *Harmonics* and the *Mixer* measurement (with option R&S ZVA-K4).

**Remote control:** `[SENSe<Ch>:]CORRection:COLLect:CONNection:PORTs ALL | SINGle`
`[SENSe<Ch>:]FREQuency:CONVersion:AWReceiver[:STATe]`

## Logical Port

Defines the properties of the logical (balanced) ports defined in the *Balanced and Measured Ports* dialog. To display this section, at least one balanced port configuration must be defined and *Logical Ports – Ref Impedance* must be selected in the *Displayed Columns* dialog.

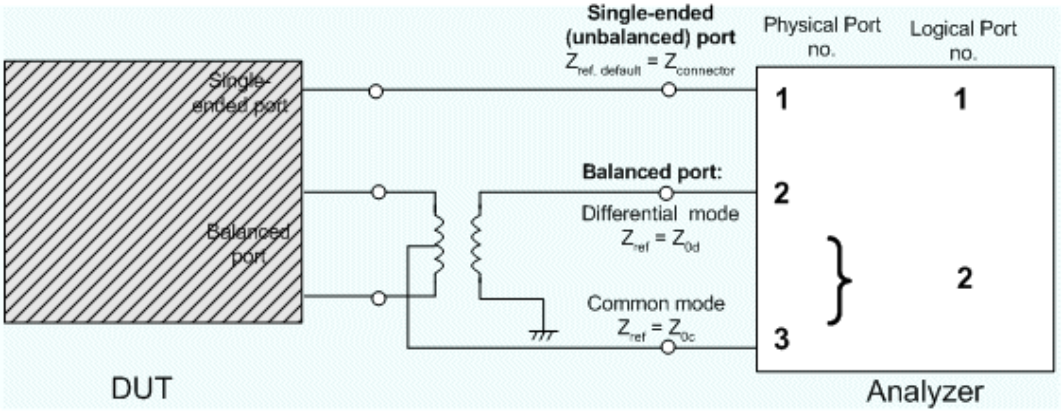



- # is the port number of the logical port. This number is independent of the physical port numbers and of the ports that are combined to form a balanced port.
- *Ref Impedance* shows the complex reference impedances of the ports.

**Reference impedance settings**

The default reference impedance for a physical port is equal to the reference impedance of the connector type assigned to the port but 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.




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.

**Remote control:**
```
SENSe:PORT<phys_port>:ZREFerence <real> [,<imaginary]
SENSe:LPORT<log_port>:ZCOMmon <real> [,<imaginary]
SENSe:LPORT<log_port>:ZDIFferent <real> [,<imaginary]
```

## Physical Port

Shows and/or defines the properties of the physical (unbalanced) ports of the analyzer.

| Physical Port | | | |
|---|---|---|---|
| # | Instrum | Connector | Ref Impedance |
| Port 1 | ZVA8 | N 50 Ω (f) | 50 Ω + j0 Ω ... |
| Port 2 | ZVA8 | N 50 Ω (f) | 50 Ω + j0 Ω ... |
| Port 3 | ZVA8 | N 50 Ω (f) | 50 Ω + j0 Ω ... |
| Port 4 | ZVA8 | N 50 Ω (f) | 50 Ω + j0 Ω ... |
| Gen 1 | SMP22 | | ... |
| Pmtr 1 | HP438A | | ... |

- # is the test port number of the analyzer as shown on the front panel. External generators and power meters configured in the *System Configuration* dialog are also shown in the list of physical ports.
- *Instrum.* shows the network analyzer type (e.g. ZVA8, ZVA24). The types of external generators and power meters are also displayed in this column.

- *Connector* opens a drop-down list to select the connector type An (f) behind the connector type denotes female connectors, an (m) denotes male connectors. Symmetric (sexless) connectors (e.g. PC7) are not labeled. User-defined connectors can be added or removed in the *Available Connector Types* dialog, which is opened from the *Channel – Calibration – Calibration Kits* dialog. at the port and its gender. If *Same Connector at All Ports* is active, the connector types at all ports (but not their gender) are always adjusted to the current selection.
- *Ref. Impedance* shows the adjustable complex reference impedances of the physical port. If a balanced port configuration is active the reference impedance settings for all ports are displayed in the *Logical Port* section.

**Reference impedance settings**

The default reference impedance for a physical port is equal to the reference impedance of the connector type assigned to the port but 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.

*To perform a system error correction, the physical port must be available as a source and receive port. This condition is met if it is selected as a measured port in the Balanced and Measured Ports dialog (Meas is on).*

To perform a system error correction, it is necessary that:

- the physical port is available as a source port
- both the measurement and the reference channel at the port can be measured

These conditions are met with the *Port Configuration* settings listed below.

| Section | Parameter | Value |
|---|---|---|
| Meas | On | on |
| Source | On | on |
| | Perm | off |
| Receiver | LO a | on |
| | LO b | on |

**Remote control:**
```
[SENSe<Ch>:]CORRection:COLLect:CONNection<port_no>
[SENSe<Ch>:]CORRection:COLLect:SCONnection<port_no>
[SENSe<Ch>:]CORRection:COLLect:CONNection:PORTs ALL | SINGle
SENSe:PORT<phys_port>:ZREFerence <real> [,<imaginary]
```

## Source (Most Settings for R&S ZVA and R&S ZVT Analyzers)

Shows and/or defines the RF generator settings for each physical port of the analyzer and for each external generator. No source settings are provided for external power meters.

*Most of the settings in this section require option ZVA-K4, Arbitrary Generator and Receiver Frequencies. This option is available for ZVA and ZVT network analyzers only.*

| Source | | | | | | | |
|---|---|---|---|---|---|---|---|
| RF Off | Gen | Frequency | Frequency Result | Power | Power Result | PCor Off | Slope |
| ☐ | ☐ | fb ... | 300 kHz ... 8 GHz | Pb ... | 0 dBm | | 0 dB/GHz |
| ☐ | ☐ | fb ... | 300 kHz ... 8 GHz | Pb ... | 0 dBm | | 0 dB/GHz |
| ☐ | ☐ | fb ... | 300 kHz ... 8 GHz | Pb ... | 0 dBm | | 0 dB/GHz |
| ☐ | ☐ | fb ... | 300 kHz ... 8 GHz | Pb ... | 0 dBm | | 0 dB/GHz |
| ☐ | ☐ | fb ... | 300 kHz ... 8 GHz | Pb ... | 0 dBm | | 0 dB/GHz |
| | | ... | | ... | | | |

- *RF Off* switches the RF signal source at the port off (box checked) or on. *Dependencies:* The column is disabled (grayed) while the RF power is switched off globally for the active channel. The settings are restored as soon as the RF power is switched on again. RF Off and Meas are independent from each other: While the RF power is switched off, the port is not necessarily excluded from the measurement, it can still be used as a receive port. On the other hand, an excluded port (Meas = Off) can still be used as an RF signal source.

  The source must be switched on to perform a system error correction or a source power calibration for the source port.

- *Gen* qualifies whether the RF signal source at the port is switched on for all partial measurements (box checked, permanent signal source) or only for the partial measurements that require the port as a drive port. Dependencies: *Gen* has no effect unless the RF source at the port is switched on *(RF Off* is not checked). A port which is excluded from the measurement *(Meas = Off)* can still be used as a permanent RF signal source. If *Gen* is on, then the source frequencies at port 1 and 2 and at port 3 and 4 are coupled; see note on coupled test ports below. *Gen* is unavailable in true differential mode.

**External generators**

An external generator always represents a permanent signal source that is switched on for all partial measurements. *Gen* qualifies whether or not the generator is actually available as a signal source in the test setup.

The analyzer provides two alternative ways of selecting a generator as a signal source:

- *Gen: On* in the *Port Configuration* dialog, especially if the generator is not assigned to a particular measured quantity or drive port.
- Selection as a source for a particular measured quantity or for a power calibration, see e.g. *More Wave Quantities.*

The two selection mechanisms are independent from each other.

**Permanent vs. intermittent RF signal source**

Switching on the signal source permanently eliminates the power settling times of the DUT but may introduce measurement inaccuracies, e.g. due to crosstalk between two ports. Therefore, the parameter must be switched off to perform a system error correction.

- *Frequency* opens an input box to define a port-specific stimulus frequency range (for frequency sweeps) or CW frequency (for power, time and *CW Mode* sweeps). In the default configuration the stimulus parameters (sweep range, *CW Frequency)* are used. The result is displayed in the *Frequency Result* column. A red *Frequency* field indicates that the hardware limits are exceeded, e.g. if the default sweep range of the analyzer is beyond the frequency range of the external generator.

***Coupled* test ports**

Test ports 1 and 2 and test ports 3 and 4 are supplied by a common generator (see block diagrams in the data sheet). If the RF signal is simultaneously fed to two coupled ports (i.e. if *Gen* is switched on so that at least one signal source is permanent), the port frequencies must be the same. For normal measurements (*Gen* switched off), this restriction does not apply because there is only one source port per partial measurement.

The frequency formula for a permanent signal source is also used for the second (coupled) test port.

- *Frequency Result* displays the current frequency range (for frequency sweeps) or CW frequency (for power, time and *CW Mode* sweeps); see stimulus parameters.
- *Power* opens an input box to define a port-specific source power range (for power sweeps) or fixed power (for frequency and CW sweeps). The result (not taking into account a possible *Slope)* is indicated in the *Power Result* column. In the default configuration the stimulus parameters (sweep range, internal source *Power)* are used.
- *Power Result* displays the current power range (for power sweeps) or fixed internal source power (for frequency, time and *CW Mode* sweeps); see *stimulus parameters.*
- *PCor Off* enables or disables a source power calibration for an individual port. This function is available only for ports where a source power calibration is available. It is identical with *Source Power Correction Off* in the Source Power Cal menu.
- *Slope* defines a linear factor to modify the port-specific source power as a function of the stimulus frequency. The value can be set for frequency sweeps only.

**Impact of the *Slope***

The channel power pb is defined by the following sweep settings:

- For a power sweep, the channel power is equal to the stimulus range.
- For a frequency or CW sweep, the channel power is equal to the fixed internal source *Power*.

The slope modifies the channel power pb so that the following port power ps (source power) is obtained:

$$ps = pb + <Slope> * \Delta f$$

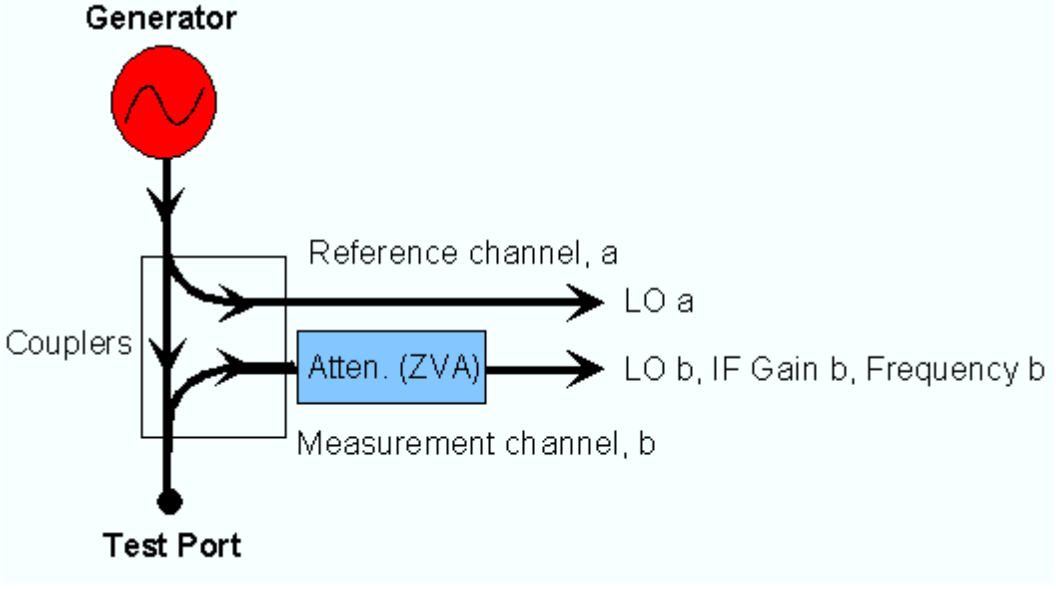
In the formula above, $\Delta$f denotes the difference between the current source port frequency and the start of the stimulus frequency range. The *Slope* parameter is not available unless a frequency sweep is active.

**Remote control:**
```
SOURce<Ch>:POWer<Pt>:STATe ON | OFF
SOURce<Ch>:POWer<Pt>:PERManent[:STATe] ON | OFF
[SENSe<Ch>:]FREQuency:STARt?
[SENSe<Ch>:]FREQuency:STOP?
[SENSe<Ch>:]SWEep:CW|FIXed?
SOURce<Ch>:POWer<Pt>[:LEVel][:IMMediate]:OFFSet
SOURce<Ch>:POWer[:LEVel][:IMMediate][:AMPlitude]?
SOURce<Ch>:POWer<Pt>:CORRection[:STATe]
SOURce<Ch>:POWer<Pt>[:LEVel][:IMMediate]:SLOPe
SOURce<Ch>:POWer<Pt>:GENerator<Gen>:OFFSet
SOURce<Ch>:POWer<Pt>:GENerator<Gen>:PERManent[:STATe]
SOURce<Ch>:POWer<Pt>[:LEVel][:IMMediate]:PHASe
```

## Receiver (Most Settings for R&S ZVA and R&S ZVT Analyzers)

Shows and/or defines the receiver settings for each physical port of the analyzer and each external power meter. No receiver settings are provided for external generators.

Some receiver settings can be defined independently for the reference channel a (the wave generated by the internal source and measured at the port) and the measurement channel b (the wave from the DUT) at each port.




*Most of the settings in this section require option ZVA-K4, Arbitrary Generator and Receiver Frequencies. This option is available for ZVA and ZVT network analyzers only.*

| Receiver | | | | | | |
|---|---|---|---|---|---|---|
| LO a | LO b | Frequency | Frequency Result | PCor Off | Atten | IF Gain b |
| ☑ | ☑ | fb ... | 300 kHz ... 8 GHz | ☐ | 0 dB | Auto |
| ☑ | ☑ | | | | 0 dB | Auto |
| ☑ | ☑ | | | | 0 dB | Auto |
| ☑ | ☑ | | | | 0 dB | Auto |
| | | ... | | | | |
| | | fb ... | 300 kHz ... 8 GHz | | | |

- *LO a/b* switch the two Local Oscillator amplifiers at the port on (box checked) or off.

  Both LOs must be switched on to perform a system error correction.

- *Frequency* opens an input box to define a converted receiver frequency range (for frequency sweeps) or CW frequency (for power, time and *CW Mode* sweeps). The converted receiver frequencies are valid for all ports. In the default configuration the stimulus parameters (sweep range, *CW Frequency)* are used. The result is indicated in the *Frequency Result* column.
- *Frequency Result* displays the current frequency range (for frequency sweeps) or CW frequency (for power, time and CW Mode sweeps); see stimulus parameters.
- *PCor Off* enables or disables a receiver power calibration for an individual port. This function is available only for ports where a receiver power calibration is available. It is identical with *Receiver Power Correction Off* in the Receiver Power Cal menu.
- *Atten* defines port-specific attenuation factors for the received wave. The settings are analogous to the global *Step Attenuator* settings and replace them.
- *IF Gain b* selects the IF gain in the receiver path.

**Receiver path settings and AGC**

The settings are valid for all ports:

- *Auto* means that the analyzer adapts the IF gain to the RF input level (Automatic or Adaptive Gain Control, AGC). The A/D converter is always operated at optimum input level.

The following settings accelerate the measurement:

- *Low Dist(ortion)* corresponds to a small IF gain(i.e. 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 Noise* corresponds to a large IF gain(i.e. a higher internal A/D converter input level). This settingincreases the dynamic range and is appropriate for low RF input levels.

The *Low Dist* or *Low Noise* settings are appropriate whenever the characteristics of the input path 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 in the vicinity of the measured signal must not overdrive the receiver.

**Remote control:**
```
[SENSe<Ch>:]LOMeasure
[SENSe<Ch>:]LOReference
[SENSe<Ch>:]FREQuency:STARt?
[SENSe<Ch>:]FREQuency:STOP?
[SENSe<Ch>:]SWEep:CW|FIXed?
[SENSe<Ch>:]CORRection:POWer[:STATe]
[SENSe<Ch>:]POWer:ATTenuation
[SENSe<Ch>:]POWer:IFGain:MEASure
```

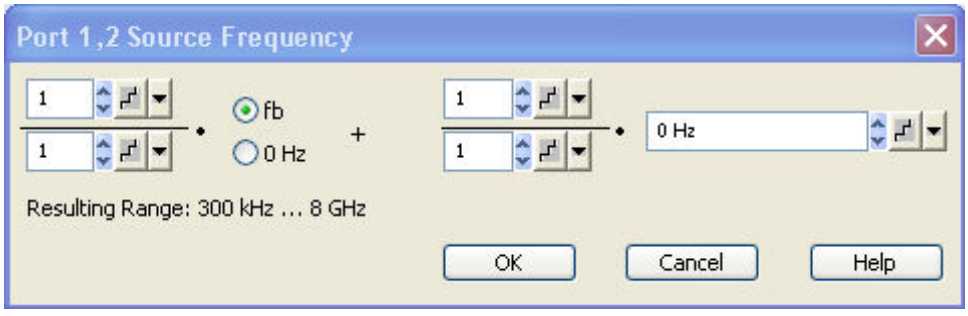
## Converted Frequencies (R&S ZVA and R&S ZVT Analyzers)

In the *Source* and *Receiver* sections of the *Port Configuration* table, it is possible to convert the source and receiver frequency ranges (for frequency sweeps) or CW frequencies (for power, time and *CW Mode* sweeps) in order to perform measurements on frequency-converting DUTs. The source frequency is port-specific whereas the receiver frequency must be the same at all ports.

NOTE

Test ports 1 and 2 and test ports 3 and 4 are supplied by a common generator (see block diagrams in the data sheet). If the RF signal is simultaneously fed to two coupled ports (i.e. if *Gen* is switched on so that at least one signal source is permanent), the port frequencies must be the same. For normal measurements *(Gen* switched off), this restriction does not apply because there is only one source port per partial measurement.
The frequency formula for a permanent signal source is also used for the second (coupled) test port.

The port frequencies are calculated as a multiple of the stimulus frequency $f_b$ plus an offset. The coefficients of the linear transformation between $f_b$ and the port frequencies are entered in dialogs of the following type:




**Relation between channel frequency and port frequencies**

The stimulus frequency $f_b$ is defined by the following sweep settings:

- For a frequency sweep or segmented frequency sweep, the channel frequency is equal to the stimulus range.
- For a power, time or *CW Mode* sweep, the channel frequency is equal to the fixed *CW Frequency*.

This linear frequency conversion formula has been modeled according to the needs of a typical frequency-converting measurement where mixer products and harmonics occur (see also the special *Mixer Mode* and *Harmonic Distortion* measurements). The port frequencies f (source frequency $f_s$or receiver frequency for the reference channel and the measurement channel fra/frb) are related to the stimulus frequency as follows:

$$f = \frac{k}{l} f_b + \frac{m}{n} c \text{ , if the fb radio button is selected}$$

$$f = \frac{m}{n} \cdot c \text{ , if the 0 radio button is selected}$$

k and m must be integer numbers, l and n nonzero positive integers. c must a multiple of 1 Hz. Moreover the total port frequency is rounded to an integer Hz-value.

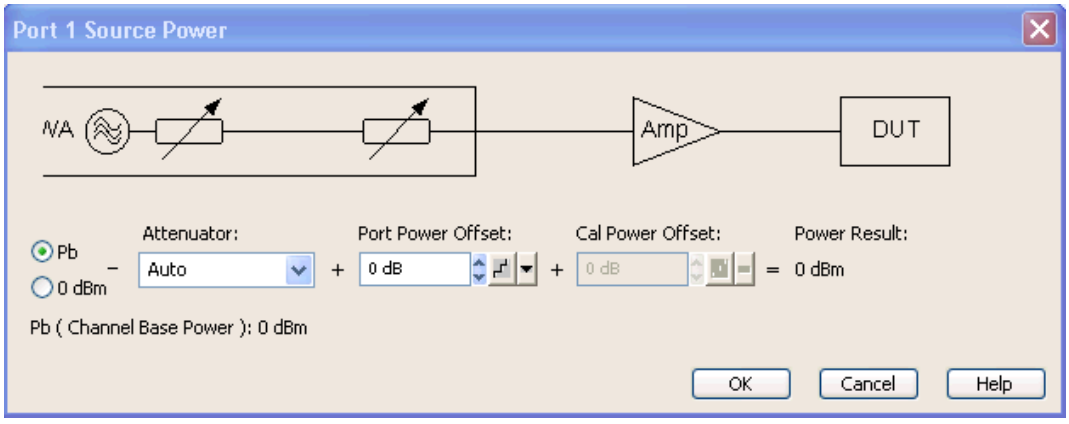
Use the Stimulus dialog to change all port frequencies based on an arbitrary reference frequency.

**Remote control:**
```
SOURce<Ch>:FREQuency<Pt>:CONVersion:ARBitrary:IFRequency
SOURce<Ch>:FREQuency<Pt>:CONVersion:ARBitrary:EFRequency<Gen>
[SENSe<Ch>:]FREQuency:CONVersion:ARBitrary
[SENSe<Ch>:]FREQuency:CONVersion:ARBitrary:PMETer<Meter>
```

## Port-Specific Power (R&S ZVA and R&S ZVT Analyzers)

In the *Source* section of the *Port Configuration* table, it is possible to define a port-specific source power range (for power sweeps) or fixed power (for frequency and CW sweeps).

The port power can be set to either the channel power $p_b$ plus an offset or to a constant value. The coefficients of the linear transformation equation between $p_b$ and the port powers are entered in a dialog.




The port power is determined by means of the following input values:

- The radio buttons labelled *Pb* or *0 dBm* specify whether the source power depends on the channel power or whether it is constant.
- *Attenuator* sets the generator step attenuator at the configured analyzer port. If *Auto* is selected, the step attenuator is set such that it doesn't have to be switched over the entire sweep. The input field is unavailable and shows *0 dB* if no generator step attenuator is installed.
- *Port Power Offset* defines a specific, constant offset for the configured analyzer port.

If a frequency sweep is active, then it is possible to add a frequency dependent offset to the source power *(Slope* setting).

*In addition to the port-specific source power the Port <n> Source Power dialog shows the Cal Power Offset. This parameter is relevant for source power calibrations with external amplifiers or attenuators and can be set via Channel – Calibration – Start Power Cal – Source Power Cal – Modify Settings – Modify Cal Power.*
*The analyzer generates a warning if the power settings for one of the ports exceed the hardware limits. At the same time, bit no. 14 in the* `. . . INTegrity:HARDware` *status register is set. Reduce or increase the power result until the warning disappears.*

**Relation between channel power and port powers**

The channel power $p_b$ is defined by the following sweep settings:

- For a power sweep, the channel power is equal to the stimulus range.
- For a frequency or CW sweep, the channel power is equal to the fixed internal source *Power*.

The port power $p_s$ (source power) is related to the channel power $p_b$ as follows:

$$ps = pb + c + <Slope> \cdot \Delta f, \text{ if the pb radio button is selected}$$
$$ps = c + <Slope> \cdot \Delta f, \quad \text{if the 0 radio button is selected}$$

In the formulas above, $\Delta f$ denotes the difference between the current source port frequency and the start of the stimulus frequency range, c is the offset value entered in the *Port <nr> Source Power* dialog minus the *Attenuator* setting. The *Slope* parameter is set in the *Source* section of the *Port Configuration* table; it is not available unless a frequency sweep is active.

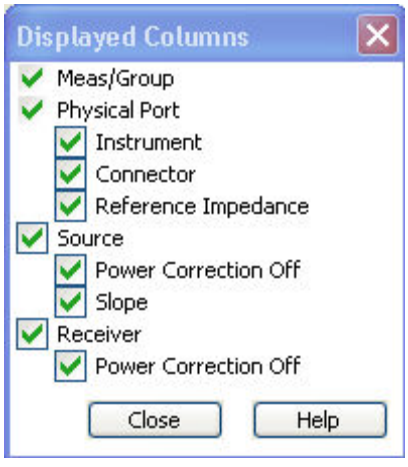
Use the *Stimulus* dialog to change all port powers based on an arbitrary reference power.

**Remote** `SOURce<Ch>:POWer<Pt>[:LEVel][:IMMediate]:OFFSet <numeric_value>,`

**control:** `ONLY | CPAD`
`SOURce<Ch>:POWer<Pt>:GENerator<Gen>:OFFSet <numeric_value>, ONLY | CPAD`

## Displayed Columns

Opens a dialog to customize the *Port Configuration* table and display or hide columns.



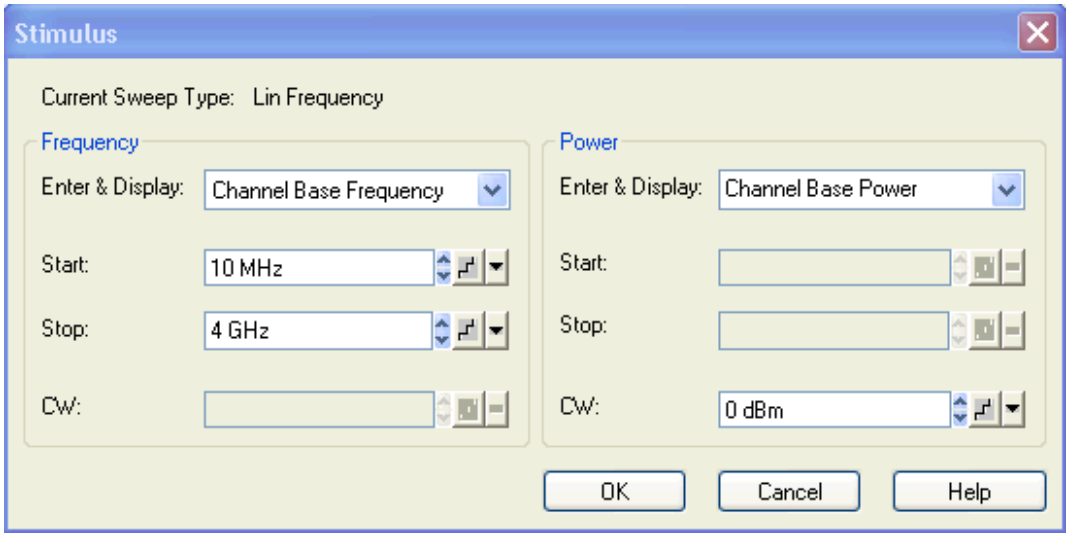

- (Un)checking *Logical Port, Source* or *Receiver* displays (removes) the corresponding sections in (from) the *Port Configuration* table.
- (Un)checking one of the second-level boxes displays (removes) the corresponding column in (from) the *Port Configuration* table.

The physical port number is always displayed. The logical port information is displayed only if a balanced port configuration is defined.

**Remote control:** No command, display configuration only.

## Stimulus Dialog

The *Stimulus* dialog modifies all sweep frequencies and powers in the active channel, based on an arbitrary frequency and power reference. This is particularly useful in arbitrary mode (with option R&S ZVA-K4), where the frequencies and source powers at the ports are independent. The dialog is accessed from the *Port Configuration* dialog.




The available sweep parameters depend on the *Current Sweep Type (Channel – Sweep – Sweep Type)* which is displayed below the title bar of the dialog.

| Sweep Type | Frequency parameters | Power parameters |
|---|---|---|
| Lin. Frequency<br>Log. Frequency | Start<br>Stop | CW |
| Segmented Frequency | – (defined by the sweep segments) | CW |
| Power | CW | Start<br>Stop |
| Time<br>CW Mode | CW | CW |

The frequencies and powers are displayed in two separate but analogous panels:

- *Enter & Display* in the *Frequency* panel contains a list of the channel base frequency $f_b$ plus the source and receiver frequencies at all logical ports that depend on $f_b$. The list includes the frequencies of external power meters and generators. Fixed source and receiver frequencies (see *Converted Frequencies*) are not in the list; they are not affected by the settings in the *Stimulus* dialog. For R&S ZVA analyzers without option R&S ZVA-K4, all frequencies and powers are coupled so that this selection has no effect.
- *Enter & Display* in the *Power* panel contains a list of the channel base power $p_b$ plus the source and receiver powers at all logical ports that depend on $p_b$. The list includes the powers of external power meters and generators. Fixed source and receiver powers (see *Port-Specific Power*) are not in the list; they are not affected by the settings in the *Stimulus* dialog.
- *Start, Stop* define the frequency/power sweep ranges; *CW* defines the fixed frequency/power.

**Frequency and power definition in the *Stimulus* dialog**

If the selected reference frequency or reference power is changed, the analyzer modifies all source and receiver frequencies/powers accordingly, leaving the frequency and power conversion formulas unchanged. The *Stimulus* dialog thus ensures that all system frequencies/powers can be modified consistently, based on an arbitrary reference frequency.

The power and frequency entries in the *Stimulus* dialog are unrestricted. A compatibility check is performed when the dialog is closed. Frequencies and powers beyond the analyzer's hardware limits

are marked with a red input field in the calling *Port Configuration* dialog.

By default, all port frequencies/powers are referenced to the base frequency/power *(Channel Base $f_b$/$P_b$)*. If another port frequency or power, which was selected as a reference, becomes invalid (e.g. because it is set to a constant value), it is automatically replaced by the channel base frequency/power.

**Remote control:** `[SENSe<Ch>:]SWEep:AXIS:FREQuency`
`[SENSe<Ch>:]SWEep:AXIS:POWer`

## Balanced and Measured Ports

Opens a dialog to select a balanced port configuration and define reference impedances for balanced ports and to select the measured ports.

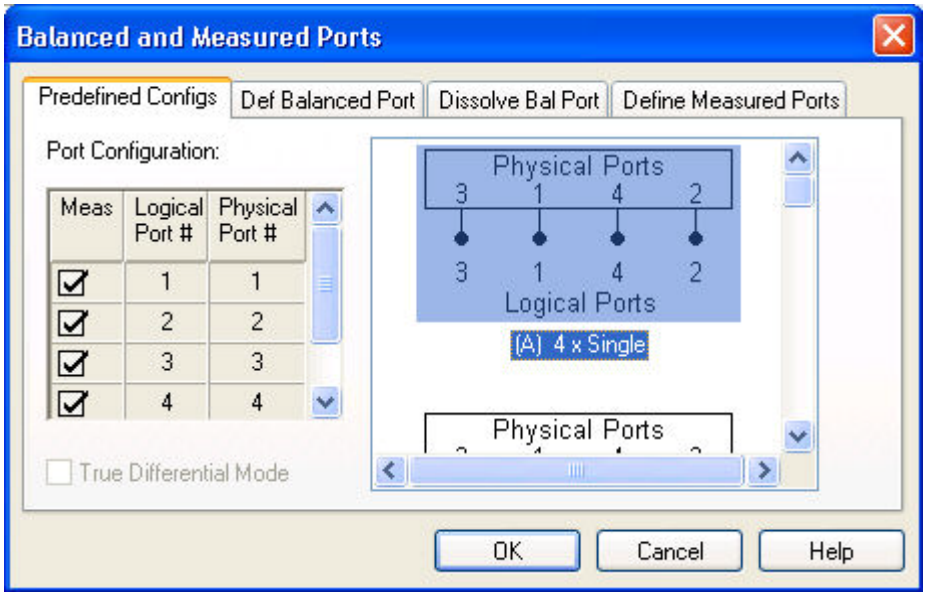
**Balanced and Measured Ports**

Selecting a balanced port configuration means that one or more pairs of physical test ports are combined to form logical (balanced) ports.

Unbalance-balance conversion is the simulation of one or more unbalance-balance transformers (baluns) integrated in the measurement circuit in order to convert the DUT ports from an unbalanced state into a balanced state and virtually separate 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. The balanced ports of the DUT are directly connected to the analyzer ports and no physical transformer is needed.

Measurements do not necessarily require all of the physical and logical ports of the network analyzer. To save measurement time, it is recommended to define the ports that are actually used in the current test setup *(Define Measured Ports)*. Unused ports will not be considered for the calculation of mixed mode, Z- and Y-parameters.

Option R&S ZVA-K6 provides the true differential mode. The true differential mode extends the virtual differential mode described so far, where the analyzer generates an unbalanced stimulus signal and uses mathematical transformations to determine the balanced results.

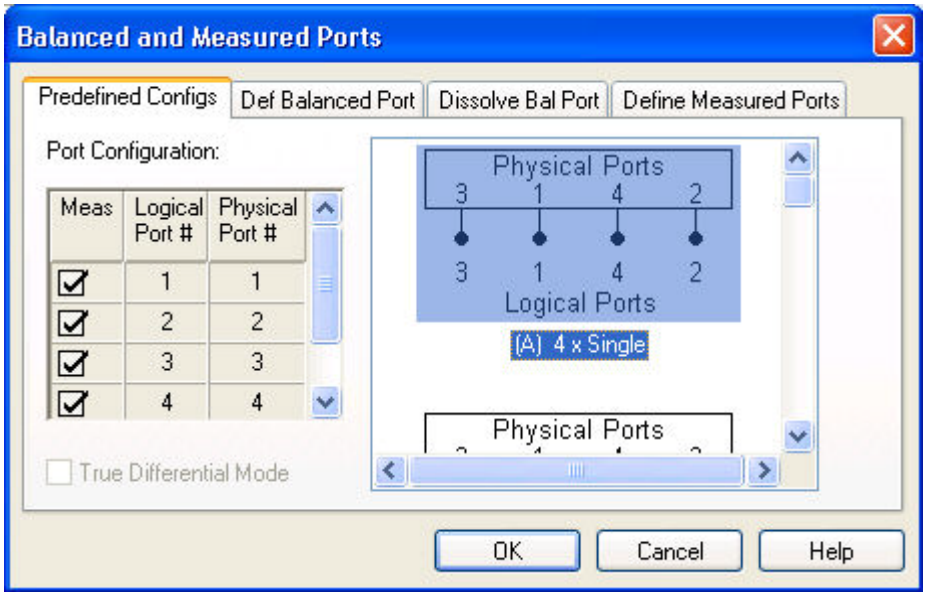



In the four tabs of the *Balanced and Measured Ports* dialog it is possible to

- Select one of the predefined balanced port configurations *(Predefined Configs).*
- Define a logical port based on two physical ports and independent reference impedances for the calculation of common mode and differential mode parameters *(Def. Balanced Port)*.
- Dissolve a single logical port so that the two involved ports are again used as physical ports *(Dissolve Bal. Port)*.
- Select the ports that can be used for the measurement *(Define Measured Ports).*
- Switch the *True Differential Mode* on or off.

## Predefined Configs

The *Predefined Configs* tab of the *Balanced and Measured Ports* dialog provides the most commonly used balanced port configurations for the analyzer.




The port configurations are arranged in a scrollable list and selected with a mouse click. The resulting

port number assignment is shown on the left-hand side of the *Balanced and Measured Ports* dialog and in the *Port Configuration* dialog.

The *True Differential Mode* (option R&S ZVA-K6) is only available if a balanced port configuration is active.

**Predefined Configurations**

The number of type of predefined configurations depends on the port number of the analyzer.

The predefined configurations comprise the fully single-ended (unbalanced) case, where the logical ports correspond to the physical ports, and all balanced configurations where two adjacent physical ports are connected to a logical port. In addition configurations where one or more ports are excluded from the measurement are provided (see *Measured Ports)*.

**Example:** 4-port analyzer

- Fully unbalanced: Physical ports no. 3, 1, 4, 2 connected to logical ports (DUT ports) 3, 1, 4, 2.
- One balanced port:

a) Physical ports no. 3 and 1 connected to logical port 1, remaining ports unbalanced.
b) Physical ports no. 4 and 2 connected to logical port 2, remaining ports unbalanced.

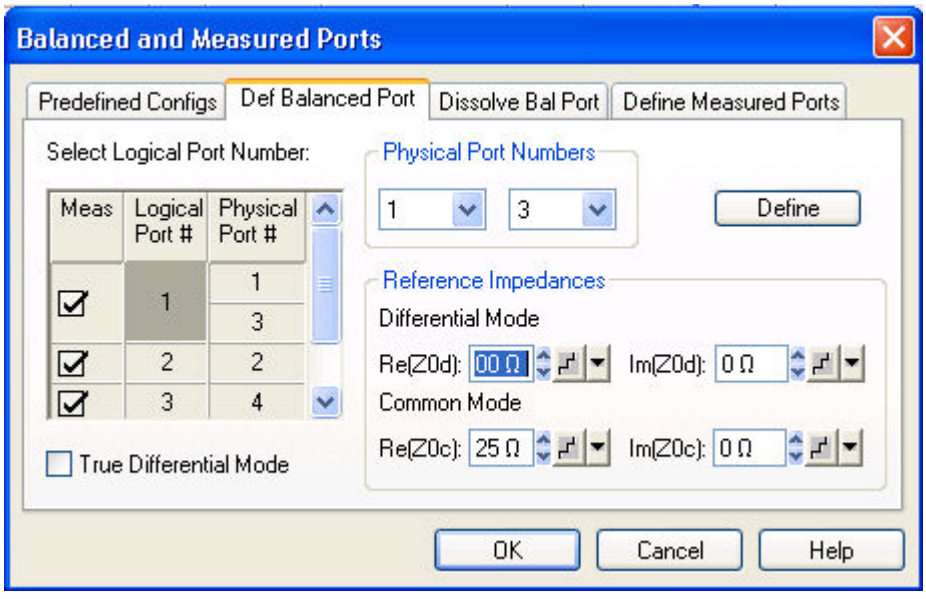
- Two balanced ports: Physical ports no. 3 and 1 connected to logical port 1, physical ports no. 4 and 2 connected to logical port 2.
- In additional configurations one, two or three ports (marked as ) are excluded from the measurement.

To define a non standard configuration or use different port impedances, select the *Def. Balanced Port* tab.

**Remote control:** `SOURce<Ch>:LPORt<log_port> <phys_port1>,<phys_port2>`
(no extra command for predefined configurations)
`SOURce<Ch>:TDIF[:STATe] ON | OFF`

## Def. Balanced Port

The *Def. Balanced Port* tab of the *Balanced and Measured Ports* dialog defines new balanced port configurations and reference impedances for common and differential mode.

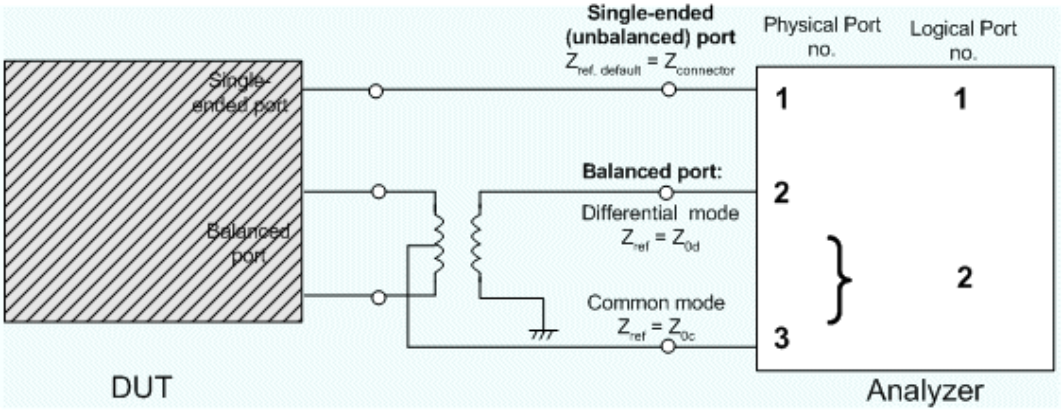



In principle, it is possible to combine any pair of two physical analyzer ports. An n-port analyzer supports a maximum of n/2 (n even) or (n – 1)/2 (n odd) logical ports.

- *Physical Port Numbers* provides two drop-down lists to select a pair of different physical ports to be connected to the logical (DUT) port.
- *Reference Impedances* provides input fields for the differential and common mode impedances.
- *Select Logical Port Number* shows the current port configuration. Clicking the table cells updates the left *Physical Port Numbers* table and makes it easier to define new logical ports.
- The *True Differential Mode* (option R&S ZVA-K6) is available if a balanced port configuration is active.

**Reference impedance settings**

The default reference impedance for a physical port is equal to the reference impedance of the connector type assigned to the port but 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.



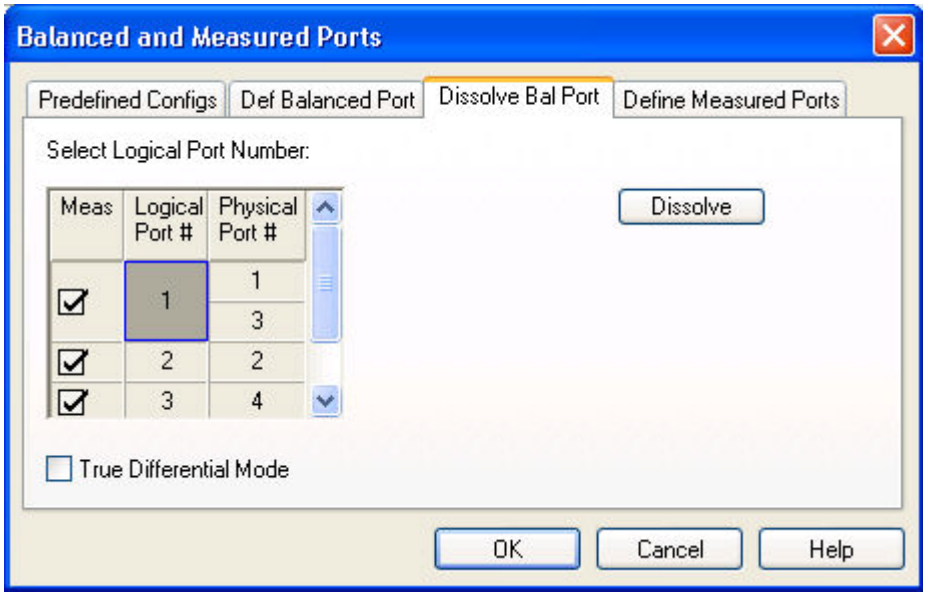

The default values for the balanced port reference impedances are derived from the (real) default reference impedance of the physical analyzer ports ($Z_0$ = 50 Ω):

- 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.

**Remote control:**
```
SOURce<Ch>:LPORt<log_port>  <phys_port1>,<phys_port2>
SENSe:LPORT<log_port>:ZCOMmon <real> [,<imaginary]
SENSe:LPORT<log_port>:ZDIFferent <real> [,<imaginary]
SOURce<Ch>:TDIF[:STATe] ON | OFF
```

## Dissolve Bal. Port

The *Dissolve* tab of the *Balanced and Measured Ports* dialog reestablishes a single-ended (unbalanced) port configuration where logical ports correspond to (single) physical ports.




- *Select Logical Port Number* shows the current port configuration. Selecting a *Logical Port #* and clicking *Dissolve* reestablishes a single-ended (unbalanced) port configuration.
- The *True Differential Mode* (option R&S ZVA-K6) is available if a balanced port configuration is active.

**Remote control:**
```
SOURce<Ch>:LPORt<log_port>:CLEar [ALL]
SOURce<Ch>:TDIF[:STATe] ON | OFF
```

## Define Measured Ports

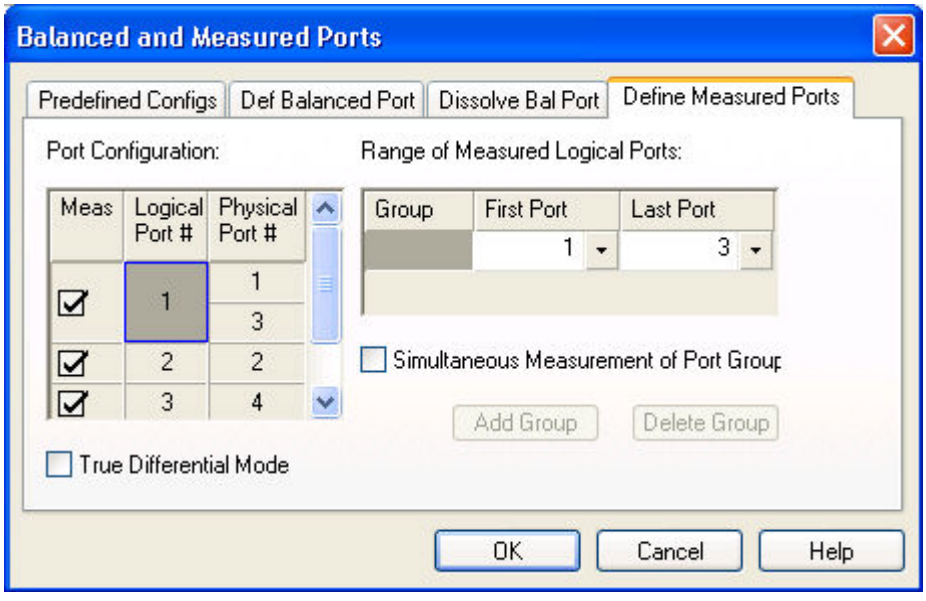
The *Define Measured Ports* tab of the *Balanced and Measured Ports* dialog selects the ports that are used for the measurement.

**Measured Ports**

Measurements do not necessarily require all of the physical or logical ports of the network analyzer. To save measurement time, it is recommended to restrict the measurement to the ports that are actually needed. Note that the analyzer will actually perform a measurement at each of the measured ports.

Unused ports will not be considered for the calculation of mixed mode, Z- and Y-parameters.

The measured ports are indicated in the first column of the *Port Configuration* table *(Meas).* The ports can be either source ports or receive ports.




- *Port Configuration* shows the current physical and balanced ports.
- *Range of Measured Ports* shows the continuous range of ports measured. Selecting another *First Port* or *Last Port* changes the range. To avoid ambiguities the selected ports are always logical ports.
- *Define* activates the current selection.
- *Simultaneous Measurement of Port Groups* activates the definition of one or more groups of ports that can be used for parallel measurements.
- The *True Differential Mode* (option R&S ZVA-K6) is available if a balanced port configuration is active.

*If a frequency-converting mode is active, port groups can not be measured in parallel.*

**Simultaneous Measurement of Port Groups**

A port group is a continuous range of measured ports which is labeled with a capital letter. The analyzer can measure several port groups in parallel. Port groups must not overlap and contain at least one channel, so the maximum number of port groups is equal to the number of ports of the analyzer.

**Example: Using port groups on a 4-port analyzer**

A 4-port analyzer can be used for parallel measurements if only 1-port, 2-port or 3-port parameters are needed:

- Combining ports 1 and 2 to a first port group, ports 3 and 4 to a second port group leaves two sets of 2-port parameters to be measured in parallel. The measurement provides 2 * 4 = 8 S-parameters and is faster than a full 4-port measurement (16 parameters).
- A reflection measurement involving four port groups, each with a single port, provides 4 parameters and is faster than the measurement with two groups of 2 ports.

The defined port groups are indicated in the first column of the *Port Configuration* table *(Meas).* Selecting *Simultaneous Measurement of Port Groups* enables two additional buttons:

- *Add Group* adds a new group (with default channels, depending on the existing port groups) to the *Port Groups* table.
- *Delete Group* deletes the selected group.

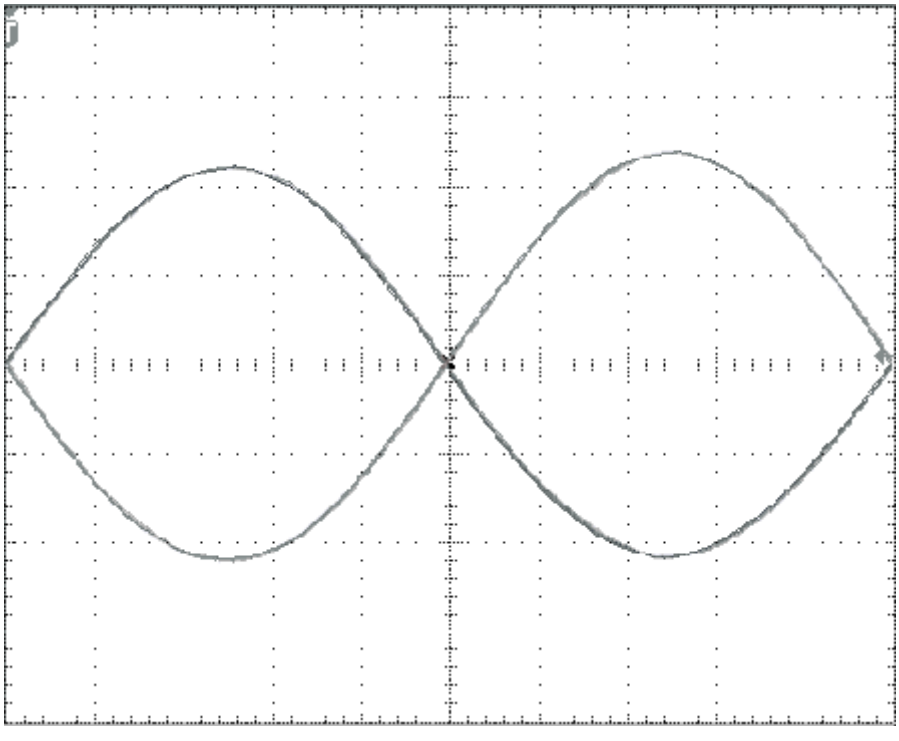
Always select the balanced port configuration **before** defining port groups. When a new balanced port is created, the analyzer deactivates *Simultaneous Measurement of Port Group* and deletes all existing port groups.

**Remote control:**
```
SOURce<Ch>:GROup<group_no>  <log_port1>,<log_port2>
SOURce<Ch>:GROup<group_no>:CLEar [ALL]
SOURce<Ch>:TDIF[:STATe] ON | OFF
```

## True Differential Mode (for R&S ZVA, all models, and R&S ZVT 8)

Differential transmission lines and circuits are widely used, because their characteristics give them a lower susceptibility to electromagnetic interference. Linear balanced devices can be tested with sufficient accuracy using the virtual differential mode, where the vector network analyzer generates unbalanced stimulus signals and uses a mathematical transformation to convert unbalanced wave quantities into balanced S-parameters. A different behavior is expected for nonlinear balanced devices, where the transmission characteristics of the DUT may depend on how closely the stimulus signal matches real operating conditions.

With option R&S ZVA-K6, True Differential Mode, the vector network analyzer can generate true differential and common mode stimuli at arbitrary reference planes in the test setup and determine mixed-mode S-parameters, wave quantities and ratios. The true differential mode also provides two additional sweep types, the amplitude imbalance and phase imbalance sweeps. Like the virtual differential mode, the true differential mode requires a port configuration with at least one balanced port. The differential mode signal that the analyzer generates in true differential mode is shown below (the plot shows the waveforms of the two single-ended sources measured vs. ground).



**Virtual and true differential mode**

The analyzer uses different stimulus signals and different mathematical methods to obtain results in normal, virtual differential, or true differential mode. The following table gives an overview.

| | |
|---|---|
| Normal (unbalanced) mode | Measurement of wave quantities and S-parameters for unbalanced ports. Unbalanced stimulus signals. |
| Virtual differential mode | Precondition: At least one balanced port is defined.<br>Measurement of unbalanced wave quantities using unbalanced stimulus signals. Unbalanced system error correction. Conversion of unbalanced wave quantities into balanced and mixed-mode S-parameters with possible renormalization of port impedances. |
| True differential mode | Precondition: At least one balanced port is defined.<br>Measurement of unbalanced wave quantities using balanced stimulus signals (differential and common mode). System error correction and conversion of unbalanced into balanced wave quantities. Calculation of balanced and mixed-mode S-parameters with possible renormalization of port impedances. |

*A consistent system error correction is essential for accurate balanced waves at the reference planes and accurate measurement results. You should always perform a full n-port calibration (TOSM, UOSM or one of the Txx calibration types) of all physical ports involved in the true differential measurement and change the differential and common mode reference impedances, if they differ from the default settings. A subsequent source power calibration is recommended. You can also use offset parameters in order to move the reference plane where the analyzer provides an accurate differential or common mode signal.*

## Source Power in True Differential Mode

A single-ended stimulus can be thought of as resulting from the superposition of a differential and a common mode stimulus, where the signals add at one single-ended port and cancel at the other (see equations in section Wave Quantities and Ratios in True Differential Mode below). Therefore the differential content of a single-ended signal is 3 dB lower than the total signal power.

The stimulus power in true differential mode can be set in two different ways:

- If *Same Differential and Common Mode Voltages as in Single-Ended Mode* is selected, the analyzer ensures that the stimulus voltage in true differential or common mode is the same as the respective stimulus components in virtual differential mode. The output power at each physical port is set to a quarter of the channel power $P_{ch}$ *(Channel – Stimulus – Power)*.
- If *Apply to Differential ($Z_d = 2Z_0$) and Common Mode ($Z_c = Z_0/2$) Waves* is selected, the channel power $P_{ch}$ is the same as the power of the true differential or common mode stimulus signal, taking their reference impedances $Z_d = 2Z_0$ and $Z_c = Z_0/2$ into account. The effect is that the output power at each physical test port that belongs to a balanced port, in reference to $Z_0$, is doubled. The channel power can be interpreted as $P_{ch} = [\text{sqrt}(2)\ U_d]^2 / Z_d = [\text{sqrt}\ (2)\ U_c]^2 / Z_c = U_d^2 / Z_0$; this is identical with the channel power in virtual differential mode $P_{ch} = U_d^2 / Z_0$.

Switching between the two stimulus power modes corresponds to a change of the channel power $P_{ch}$ by ± 3 dB.

| Mode | Stimulus signal | Nodal generator powers and voltages at physical ports |
|---|---|---|

| | | Port i | Port j |
|---|---|---|---|
| Virtual differential mode | Unbalanced | $P_{ch}$( = $U_d^2$ / $Z_0$) $U_d$ /2 + $U_c$ (= $U_d$)*) | 0 W $U_d$ /2 – $U_c$ (= 0)*) |
| True differential mode, "Same Differential and Common Mode Voltages as in Single-Ended Mode" | Differential mode | $P_{ch}$ / 4 $U_d$ / 2 | $P_{ch}$ / 4 -$U_d$ / 2 |
| | Common mode | $P_{ch}$ / 4 $U_c$ | $P_{ch}$ / 4 $U_c$ |
| True differential mode, "Apply to Differential ($Z_d$ = 2$Z_0$) and Common Mode ($Z_c$ = $Z_0$/2) Waves" | Differential mode | $P_{ch}$ / 2 $U_d$ / sqrt(2) | $P_{ch}$ / 2 -$U_d$ / sqrt(2) |
| | Common mode | $P_{ch}$ / 2 $U_c$ * sqrt(2) | $P_{ch}$ / 2 $U_c$ * sqrt(2) |

*) This entails the condition $U_d = 2 * U_c$

| Mode | Stimulus signal | Modal generator powers and voltages at balanced port | |
|---|---|---|---|
| | | Differential mode | Common mode |
| Virtual differential mode | Unbalanced | $P_{ch}$ / 2 $U_d$ | $P_{ch}$ / 2 $U_c$ |
| True differential mode, "Same Differential and Common Mode Voltages as in Single-Ended Mode" | Differential mode | $P_{ch}$ / 2 $U_d$ | 0 W 0 V |
| | Common mode | 0 W 0 V | $P_{ch}$ / 2 $U_c$ |
| True differential mode, "Apply to Differential ($Z_d$ = 2$Z_0$) and Common Mode ($Z_c$ = $Z_0$/2) Waves" | Differential mode | $P_{ch}$ $U_d$ *sqrt(2) | 0 W 0 V |
| | Common mode | 0 W 0 V | $P_{ch}$ $U_c$ *sqrt(2) |

**Avoiding possible problems at very large or very low source powers**

**1st scenario: Large source powers**

The maximum output power is not always exactly the same for different physical test ports. The deviations are generally small (< 1 dB).

If the network analyzer in true differential mode is operated in the vicinity of the maximum channel power (> +5 dBm), a message "Port<n> power unleveled" may indicate that one of two combined physical ports cannot provide the required source power. The measurement is not aborted, however, the analyzer is no longer capable of providing accurate balanced waves.

If you operate the analyzer close to its maximum power, first check the reference channels (a-waves) and ensure a 1 dB reserve over the entire sweep range.

**2nd scenario: Small source powers**

To generate equal source powers, the signal-to-noise ratio at each port must be sufficiently high. At very low levels (<–30 dBm) and large IF bandwidths, a message "Port<n> power unleveled" may indicate that the signal-to-noise ratio at one of the ports is too low. Again, the measurement is not

aborted, however, the analyzer is no longer capable of providing accurate balanced waves.

If you operate the analyzer at very low output power settings, reduce the IF bandwidth to 1 kHz or below.

The source power mode is defined in the *General* tab of the *System Configuration* dialog. It is a global setting which is not affected by a reset of the instrument.

**Remote control:** `SOURce<Ch>:TDIF:WAVes SENDed | DCMode`

## Wave Quantities and Ratios in True Differential Mode

In true differential mode, the analyzer can determine the balanced wave quantities and ratios for all balanced ports. The balanced quantities appear in the *More Wave Quantities* or *More Ratios* dialog as soon as the true differential mode is active.

Assume that a balanced port numbered i comprises the two physical ports k and l. To obtain the balanced wave quantities for port i, the analyzer measures the unbalanced wave quantities at ports k and l. The differential mode waves $a_{di0}$ and $b_{di0}$ and the common mode waves $a_{ci0}$ and $b_{ci0}$ can be calculated from the unbalanced waves using the following equations:

$$a_{di0} = \frac{1}{\sqrt{2}}(a_k - a_l)\,;\quad b_{di0} = \frac{1}{\sqrt{2}}(b_k - b_l)$$

$$a_{ci0} = \frac{1}{\sqrt{2}}(a_k + a_l)\,;\quad b_{ci0} = \frac{1}{\sqrt{2}}(b_k + b_l)$$

**Possible modification for amplitude and phase imbalance sweep**

In an amplitude imbalance or phase imbalance sweep, it is possible to compensate the a waves for the known phase imbalance $\phi$ and the amplitude imbalance $r = |\, a_k / a_l \,|$. This modifies the formulas for the balanced wave quantities as shown below.

For an amplitude imbalance sweep with known imbalance r:

$$a_{ci0} = \frac{1}{\sqrt{2}}\left(a_k \frac{1}{\sqrt{r}} + a_l \sqrt{r}\right)\,;\quad a_{di0} = \frac{1}{\sqrt{2}}\left(a_k \frac{1}{\sqrt{r}} - a_l \sqrt{r}\right)$$

For a phase imbalance sweep with known imbalance $\phi$:

$$a_{di0} = \frac{1}{\sqrt{2}}\left(a_k - a_l \cdot e^{-j\phi}\right)\,;\quad a_{ci0} = \frac{1}{\sqrt{2}}\left(a_k + a_l \cdot e^{-j\phi}\right)$$

The amplitude and phase imbalances are known quantities (sweep parameters). The unbalanced waves $a_k$ and $a_l$ are measured in the reference channels. With ideal unbalanced waves and no additional disturbing effects, the compensated balanced stimulating a-waves remain constant over the entire sweep range.

**Renormalization of port impedances**

In the default scenario where the reference impedances for the differential and common mode are equal to $Z_d = 2\ Z_0$ and $Z_c = 1/2\ Z_0$, the waves $a_{di0}$, $b_{di0}$, $a_{ci0}$ and $b_{ci0}$ correspond to the true balanced waves at port i. With arbitrary reference impedances for the balanced waves, an additional

renormalization step is necessary:

$$\begin{pmatrix} a_{di} \\ b_{di} \end{pmatrix} = \frac{1}{4Z_0} \left| \frac{2Z_0}{Z_d} \right| \sqrt{\frac{\mathrm{Re}(Z_d)}{2\,\mathrm{Re}(Z_0)}} \cdot \begin{pmatrix} 2Z_0 + Z_d & 2Z_0 - Z_d \\ 2Z_0 - Z_d & 2Z_0 + Z_d \end{pmatrix} \cdot \begin{pmatrix} a_{di0} \\ b_{di0} \end{pmatrix}$$

$$\begin{pmatrix} a_{ci} \\ b_{ci} \end{pmatrix} = \frac{1}{Z_0} \left| \frac{Z_0}{2Z_c} \right| \sqrt{\frac{2\,\mathrm{Re}(Z_c)}{\mathrm{Re}(Z_0)}} \cdot \begin{pmatrix} \frac{Z_0}{2} + Z_c & \frac{Z_0}{2} - Z_c \\ \frac{Z_0}{2} - Z_c & \frac{Z_0}{2} + Z_c \end{pmatrix} \cdot \begin{pmatrix} a_{ci0} \\ b_{ci0} \end{pmatrix}$$

The reference impedances $Z_c$ and $Z_d$ can be entered in the *Balanced and Measured Ports* dialog.

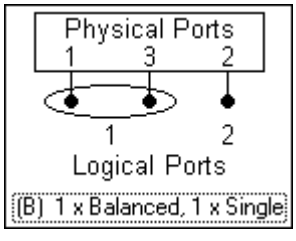
**Remote control:** `CALCulate<Ch>:PARameter:SDEFine 'AS1D2S' |`

## S-Parameters in True Differential Mode

The mixed-mode S-matrix elements that the analyzer acquires in true differential mode correspond to the mixed-mode matrix elements obtained in virtual differential mode, however, the analyzer uses true differential and true common mode stimuli at each balanced port of the DUT.

**Example: 3x3 mixed-mode S-matrix**

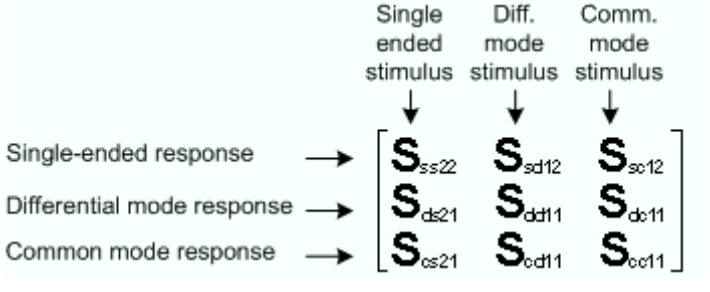
Suppose that a three-port analyzer is configured for one balanced and one single-ended port as shown below:




To obtain the complete mixed-mode S-matrix, the analyzer generates the following stimulus signals:

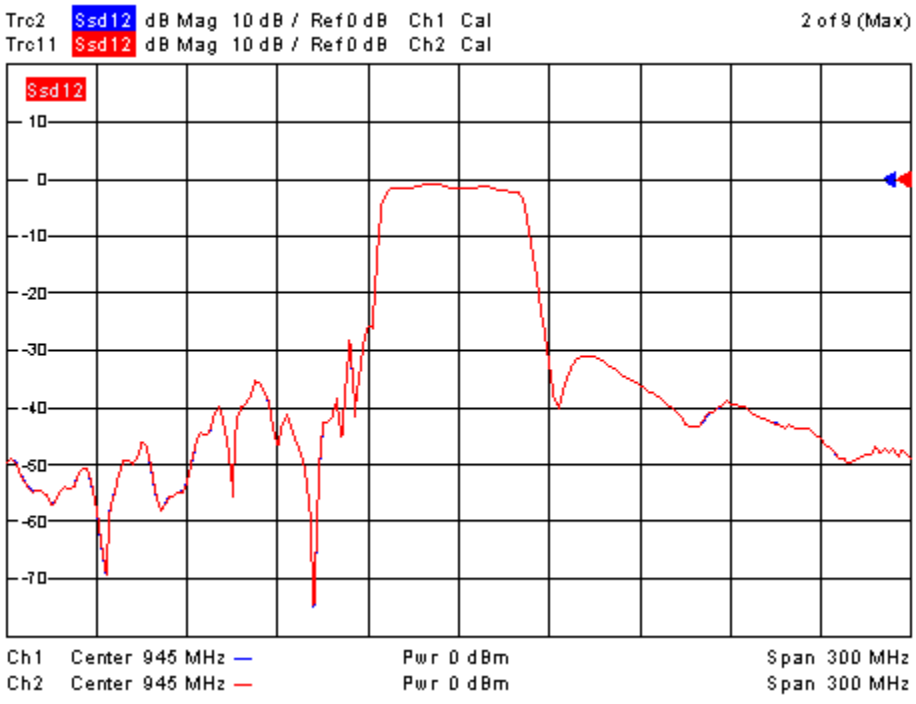
1. Differential mode signal fed to balanced port no. 1 of the DUT
2. Common mode signal fed to balanced port no. 1 of the DUT
3. Unbalanced signal fed to single-ended port no. 2 of the DUT

The DUT is fully characterized by the following mixed mode matrix:

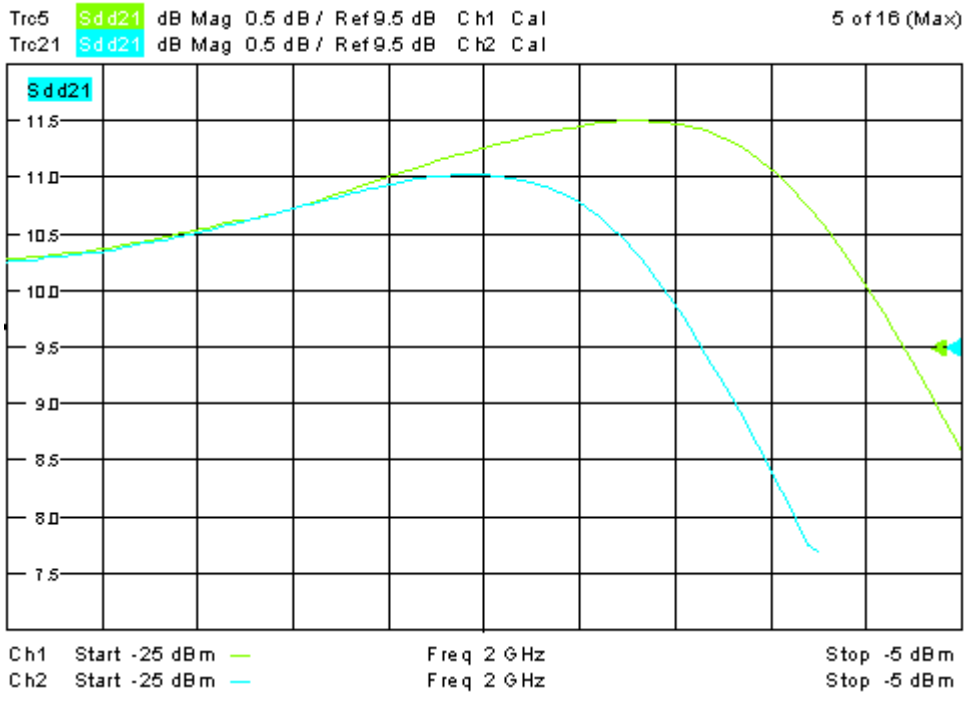



For linear DUTs, the S-matrices acquired in virtual and in true differential mode are expected to be equal. The following figure shows a comparison for the transmission coefficient $S_{sd12}$. The red trace was measured in true differential mode. The blue trace (below, measured in virtual differential mode) is

almost identical over the entire sweep range.




Differences may appear for nonlinear devices at high stimulus power levels. For example, the bias of semiconductor devices like transistors may depend on the kind of stimulus signal. The following figure shows a power sweep measured in virtual differential mode (green) and in true differential mode (blue). In true differential mode, gain peaking is smaller and compression starts at much lower stimulus power.




**Remote control:** `CALCulate<Ch>:PARameter:SDEFine 'SCD11' |`

## Amplitude Imbalance Sweep

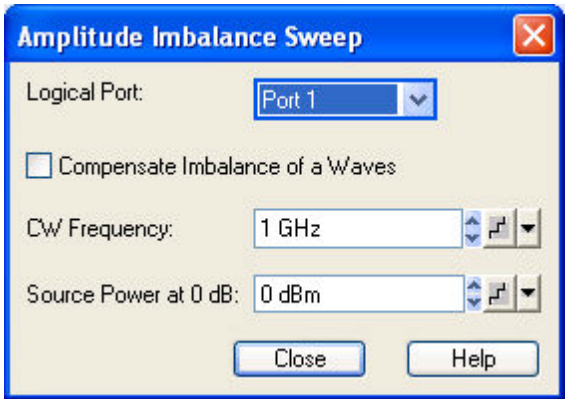
The amplitude imbalance sweep is a special application of the true differential mode. The analyzer generates a balanced signal at one of its logical ports, however, the amplitude relation of the signal components is varied according to the selected power sweep range. This variation may have an impact on the measured S-parameters if the DUT is operated in its nonlinear range.

**Characteristics of the sweep**

The amplitude imbalance sweep is performed as follows:

- The amplitude imbalance is defined as r dB = 20 log ($|a_k/a_l|$) dB, where $a_k$ and $a_l$ are the wave quantities of the physical ports with the higher and lower port numbers.
- The selected amplitude imbalance is applied to both differential and common mode signals. The analyzer generates both stimulus modes according to what the measurement of the selected quantity requires.
- The powers of the two sources are swept symmetrically around the equilibrium state $|a_k| = |a_l|$, where r dB = 0 dB (see also *Wave Quantities and Ratios in True Differential Mode)*.

At least one logical port must be defined, and the true differential mode must be active to enable the amplitude imbalance sweep. Activating the sweep *(Channel – Sweep – Sweep Type: Amplitude Imbalance)* opens the following configuration dialog:




- *Logical Port* selects the swept logical port.
- *Compensate Imbalance of a Waves* selects the calculation method for S-parameters, ratios and derived quantities; see background information below.
- *CW Frequency* defines the frequency of all stimulus signals, which is constant over the entire sweep. This setting is identical with the CW frequency for power, time and CW mode sweeps.
- *Source Power at 0 dB* defines a constant power reference, corresponding to the channel power $P_{ch}$ in the tables in section *Source Power in True Differential Mode.* The swept power range *(Channel – Stimulus – Start, Channel – Stimulus – Stop)* is defined relative to the reference value. This setting is identical with the constant power for frequency, time and CW mode sweeps.

The power range for the amplitude imbalance sweep is independent of the power range for power sweeps.

**Compensate Imbalance of a Waves**

In the default setting, the analyzer uses the differential and common mode waves, including the imbalance, to calculate S-parameters, ratios, and derived quantities. The S-parameters depend solely on the properties of the DUT. For linear DUTs, the effect of the imbalance parameters cancels so that

the S-parameters remain constant over the entire sweep range.

If *Compensate Imbalance of a Waves* is selected, the analyzer uses a modified set of stimulating a-waves to calculate S-parameters, ratios, and derived quantities; see *Wave Quantities and Ratios in True Differential Mode*. The modification compensates the effect of the imbalance parameters on the a-waves of the driving ports. As a result, the differential and common mode a-waves are approximately constant across the entire sweep. As the actual stimulus waves are imbalance-dependent, the S-parameters (and other measured quantities) show the effect of the imbalance parameters.

**Remote control:**
```
[SENSe<Ch>:]SWEep:TYPE IAMPlitude
SOURce<Ch>:TDIF:IMBalance:AMPLitude:LPORt
SOURce<Ch>:TDIF:IMBalance:AMPLitude:STARt
SOURce<Ch>:TDIF:IMBalance:AMPLitude:STOP
CALCulate<Ch>:TDIF:IMBalance:COMPensation[:STATe]
```

## Phase Imbalance Sweep

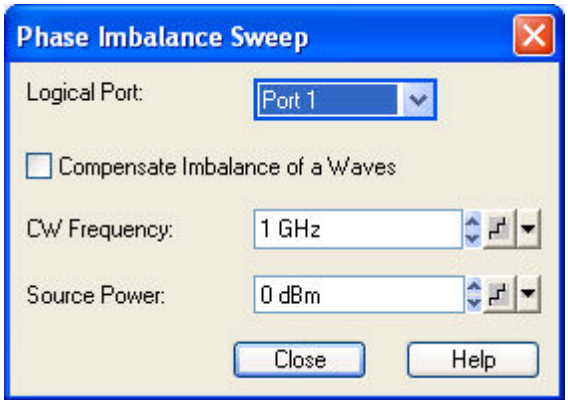
The phase imbalance sweep is a special application of the true differential mode. The analyzer generates a balanced signal at one of its logical ports, however, the relative phase of the two signal components is varied according to the selected phase range. This variation may have an impact on the measured mixed-mode S-parameters if the DUT is operated in its nonlinear range.

**Characteristics of the sweep**

The phase imbalance sweep is performed as follows:

- The phase imbalance of the differential mode signal is defined as the difference in phase (in deg) of the wave quantities at the two physical ports of the swept logical port plus 180 deg: Imbalance = arg ($a_k$) – arg ($a_l$) – 180 deg, where k > l.
- The phase imbalance of the common mode signal is defined as the difference in phase (in deg) of the wave quantities at the two physical ports of the swept logical port: Imbalance = arg ($a_k$) – arg ($a_l$), where k > l.
- The selected phase imbalance is applied to the differential as well as to the common mode signal. The analyzer generates both stimulus modes according to what the measurement of the selected quantity requires.

At least one logical port must be defined, and the true differential mode must be active to enable the phase imbalance sweep. Activating the sweep *(Channel – Sweep – Sweep Type: Phase Imbalance)* opens the following configuration dialog:




- *Logical Port* selects the swept logical port.

- *Compensate Imbalance of a Waves* selects the calculation method for S-parameters, ratios and derived quantities; see background information in section *Amplitude Imbalance Sweep*.
- *CW Frequency* defines the frequency of all stimulus signals, which is constant over the entire sweep. This setting is identical with the CW frequency for power, time and CW mode sweeps.
- *Source Power* defines the constant power of the balanced source, corresponding to the channel power $P_{ch}$ in the tables in section *Source Power in True Differential Mode*. This setting is identical with the constant power for frequency, time and CW mode sweeps.

The range of relative phases for the phase imbalance sweep is set via *Channel – Stimulus – Start* and *Channel – Stimulus – Stop*.

**Remote control:**
```
[SENSe<Ch>:]SWEep:TYPE IPHase
SOURce<Ch>:TDIF:IMBalance:PHASe:LPORt
SOURce<Ch>:TDIF:IMBalance:PHASe:STARt
SOURce<Ch>:TDIF:IMBalance:PHASe:STOP
CALCulate<Ch>:TDIF:IMBalance:COMPensation[:STATe]
```

## Harmonics

The *Harmonics* submenu controls the measurement of arbitrary harmonics or ratios between harmonics and the fundamental wave. Together with the *Mixer* measurement and the arbitrary configuration of generator and receiver frequencies *(Port Configuration),* the harmonics measurement is included in option R&S ZVA-K4.

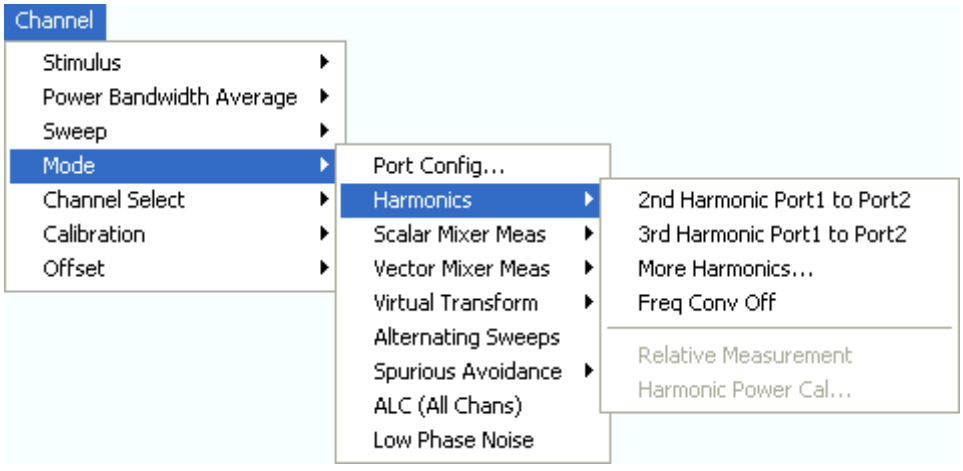
**Harmonics Measurements**

Harmonics are signals at an integer multiple of the fundamental frequency. The fundamental is the first harmonic, the $n^{th}$ harmonic is n times the frequency of the fundamental. The production of harmonic frequencies by an electronic system when a signal is applied at the input is known as harmonic distortion.

The purpose of the *Harmonics* menu is to measure the harmonic distortion of a DUT. To this end the source remains at the fundamental frequency whereas the receiver is set to n times the fundamental frequency. Two different types of results are provided:

- In the direct measurement, the $n^{th}$ harmonic of the stimulus signal is measured. The result is displayed as a wave quantity but can be modified and converted using the functions in the *Trace* menu.
- In the relative measurement, the $n^{th}$ harmonic of the stimulus signal is divided by $1^{st}$ harmonic (fundamental) received from the DUT. The result corresponds to the $n^{th}$ harmonic distortion factor.

Harmonic distortion measurements can be performed irrespective of the sweep type: A frequency sweep yields the harmonics as a function of the frequency, a power sweep yields the power-dependent harmonics at fixed frequency.



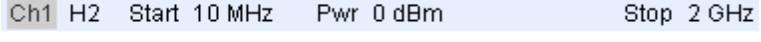

- *2nd Harmonic Port 1 to Port 2...* activates the measurement of the second harmonic at port 2 with port 1 as source port.
- *3rd Harmonic Port 1 to Port 2...* activates the measurement of the third harmonic at port 2 with port 1 as source port.
- *More Harmonics* opens a dialog for higher harmonics and arbitrary source and receive ports.
- *Freq Conv Off* activates the measurement of the fundamental with the current port assignments and trace settings.
- *Relative Measurement* calculates the ratio of the selected harmonic to the fundamental and displays the result as a new trace.
- *Harmonic Power Cal...* opens a dialog to perform a power calibration of the signal source and of the receiver.

NOTE

*The result of several harmonic measurements can be displayed in a common diagram area (see Traces, Channels and Diagram Areas). To make the comparison of the fundamental trace and the harmonics easier the stimulus axis for frequency sweeps remains unchanged, and all fundamentals are displayed as a function of the stimulus frequency (rather than the harmonic frequencies).*
*A warning Frequency out of range. Please check dialog Mode: Port Configuration. is displayed if one of the harmonic frequencies to be measured exceeds the frequency range of the analyzer. The harmonic traces are truncated at the maximum receiver frequency.*
*The harmonic measurement mode is cancelled if the channel settings (e.g. port assignments, port frequencies) are no longer compatible with the selected harmonic.*

## 2nd Harmonic Port1 to Port2

Activates the measurement of the second harmonic at port 2 with port 1 as source port. The active trace shows the wave quantity $b_2$. It is possible, however, to display other quantities and change the trace settings. A label *H2* in the channel list indicates that the second harmonic measurement is active.



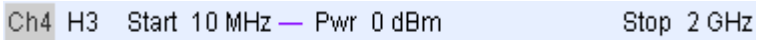

In a frequency sweep the 2nd harmonic is displayed as a function of the stimulus frequency.

**Remote control:** `[SENSe<Ch>:]FREQuency:CONVersion:HARMonic:ORDer 2`
`[SENSe<Ch>:]FREQuency:CONVersion:HARMonic:RPORt 2`
`[SENSe<Ch>:]FREQuency:CONVersion:HARMonic:SPORt 1`

Alternative: `[SENSe<Ch>:]FREQuency:CONVersion SHARmonic`

### 3rd Harmonic Port1 to Port2

Activates the measurement of the third harmonic at port 2 with port 1 as source port. The active trace shows the wave quantity $b_2$. It is possible, however, to display other quantities and change the trace settings. A label *H3* in the channel list indicates that the third harmonic measurement is active.

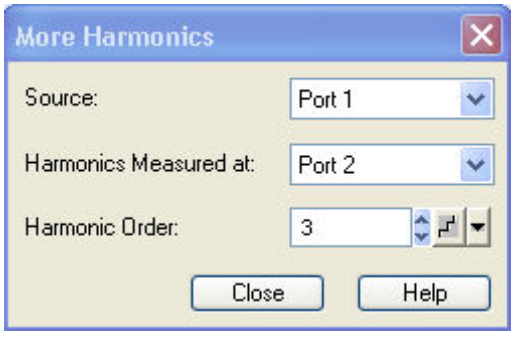



In a frequency sweep the 3rd harmonic is displayed as a function of the stimulus frequency.

**Remote control:**
```
[SENSe<Ch>:]FREQuency:CONVersion:HARMonic:ORDer 3
[SENSe<Ch>:]FREQuency:CONVersion:HARMonic:RPORt 2
[SENSe<Ch>:]FREQuency:CONVersion:HARMonic:SPORt 1
```
Alternative:
```
[SENSe<Ch>:]FREQuency:CONVersion THARmonic
```

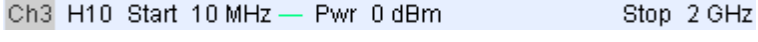
### More Harmonics...

Opens a dialog for higher harmonics and arbitrary source and receive ports.




- The *Source* port provides the stimulus signal at the fundamental frequency.
- The receive port *(Harmonics Measured at:)* measures the signal at n times the fundamental frequency where n is the *Harmonic Order*.
- The *Harmonic Order* can be set in the range between 2 (2nd harmonic) and 100.

A label *H<n>* in the channel list indicates that the nth harmonic measurement is active.




In a frequency sweep all harmonics are displayed as a function of the stimulus frequency.

*Use 2nd Harmonic Port1 to Port2 and 3rd Harmonic Port1 to Port2 to activate the measurement of the second and third harmonics with a single menu command.*

**Remote control:**
```
[SENSe<Ch>:]FREQuency:CONVersion:HARMonic:ORDer <order>
[SENSe<Ch>:]FREQuency:CONVersion:HARMonic:RPORt <port_no>
[SENSe<Ch>:]FREQuency:CONVersion:HARMonic:SPORt <port_no>
```

### Freq Conv Off

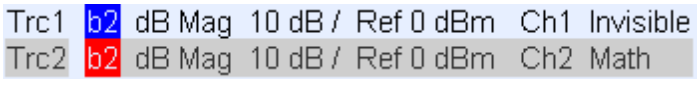
Activates the measurement of the fundamental with the current port assignments and trace settings.

This disables the harmonics measurement and all other frequency-converting measurements (e.g. scalar mixer measurements) for the active channel.

**Remote control:** `[SENSe<Ch>:]FREQuency:CONVersion FUNDamental`

## Relative Measurement

Calculates the ratio of the selected harmonic to the fundamental (harmonic distortion factor) and displays the result as a new trace. To avoid confusion between the original harmonic and the relative measurement, the harmonic is hidden (invisible) but can be retrieved using the Trace Manager.




The relative trace belongs to a new channel. In the *dB Mag* trace format, the trace points are dB values, after conversion to the lin Mag format, they correspond to the (dimensionless) ratio of two voltages:

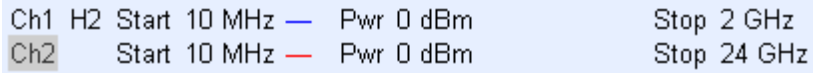
$$\text{Lin Mag}\,(Relative) = \frac{|\text{U}\,(Harmonic)|}{|\text{U}\,(Fundamental)|}$$

**Calculation of the Relative Trace**

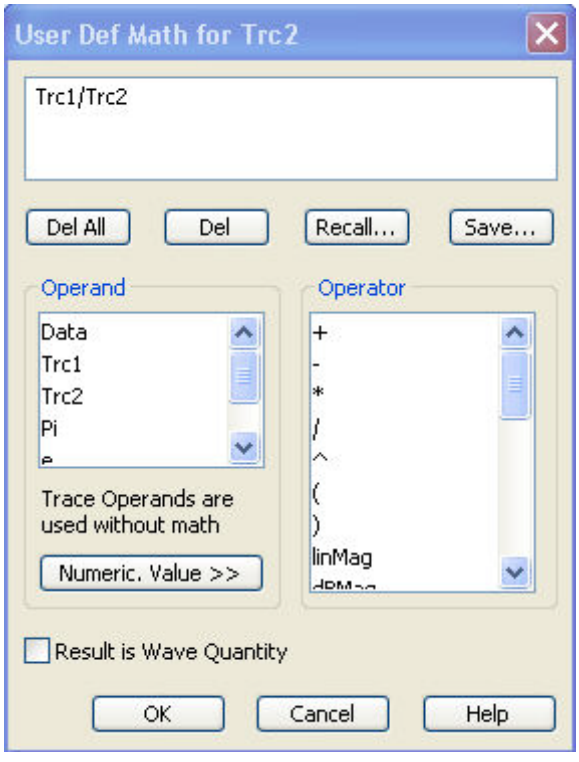
To obtain the relative trace the analyzer must measure the fundamental signal in addition to the active harmonic. In a frequency sweep the two measurements require different sweep ranges so that the fundamental is measured in a new channel. The relative trace is then calculated using trace mathematics.

Assume that the active 2nd harmonic belongs to channel Ch1 with a sweep range between 10 MHz and 2 GHz. The relative measurement involves the following steps:

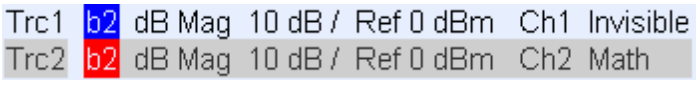
5. The analyzer creates a new channel Ch2 with maximum sweep range, e.g. for a 24 GHz analyzer:




6. The harmonic trace is measured with Ch1 settings, the fundamental trace is measured with Ch2 settings.
7. The analyzer enables the trace mathematics and divides the harmonic trace by the fundamental trace:




8. The harmonic trace is hidden (invisible); the relative trace is displayed as a mathematical trace.




**Remote control:** `[SENSe<Ch>:]FREQuency:CONVersion:HARMonic:RELative ON`

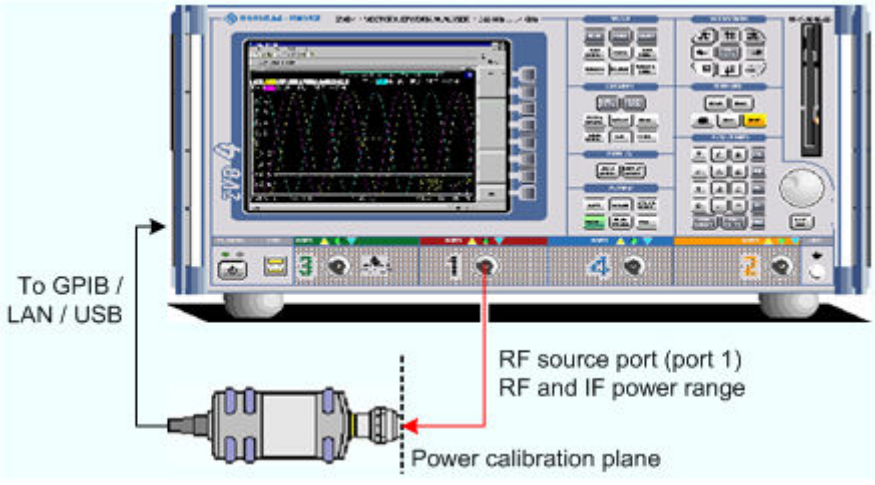
## Harmonic Power Cal

Opens a dialog to perform a power calibration of the signal source and of the receiver. A power calibration ensures accurate levels at the input of the DUT (source power calibration) and an accurate power measurement of the harmonic (receiver calibration). The necessary calibration steps are automatically performed across the entire frequency range of the active harmonic measurement.

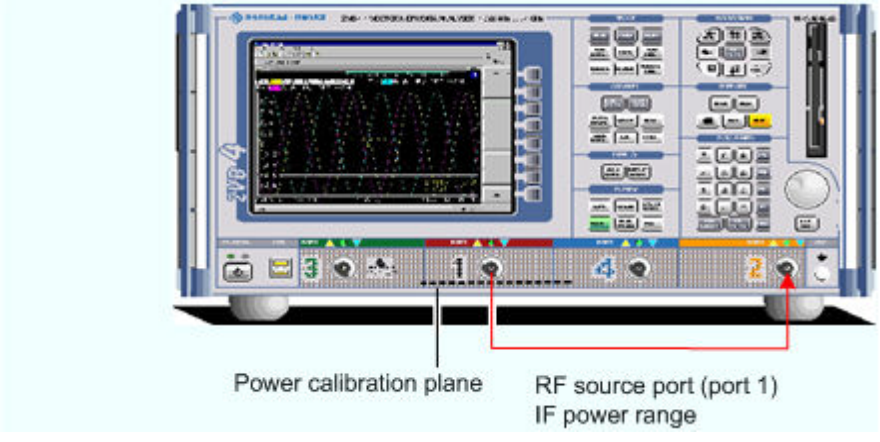
**Harmonic Power Cal procedure and example**

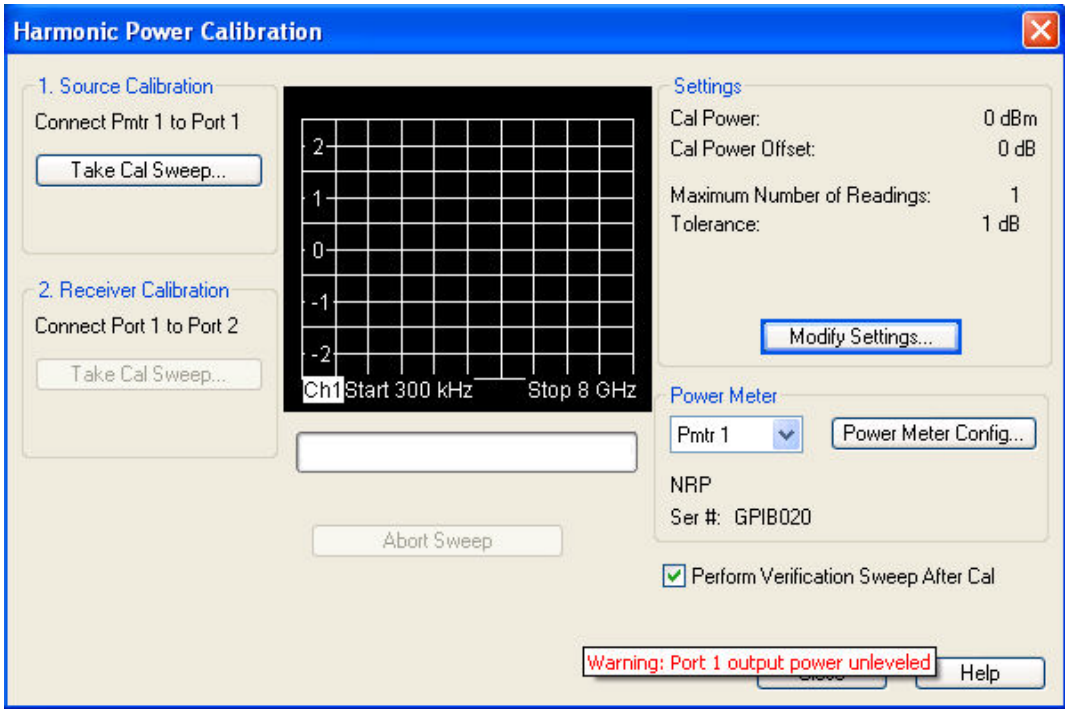
Due to the different frequency ranges of the input and output signals, the harmonic power calibration must be performed in two steps. For a test setup where the DUT's input signal (fundamental signal) is provided by analyzer port 1 and where the harmonic is analyzed at port 2, the following test setups are required:

9. Source power calibration for the fundamental signal. A power sensor is connected to port 1. The RF power (red) is calibrated over the selected sweep range.




10. Receiver power calibration for the selected harmonic. No external device is needed. The receiver (port 2) is calibrated at the IF frequency using the source signal from port 1 (red) calibrated in the first step.






The calibration procedure is controlled by means of the two buttons on the left side.

- *Take Cal Sweep* in the two numbered panels start the calibration sweeps for the source and receiver calibration, respectively. The calibration sweeps are performed at the test settings shown on the right side (see also *Source Power Cal* and *Receiver Power Cal* dialogs). The progress of the calibration is shown in the calibration sweep diagram and the messages below. *Abort Sweep* aborts the calibration.

The control elements in the right part of the dialog give access to various settings:

- *Modify Settings...* opens the *Modify Source Power Cal Settings* dialog.
- *Power Meter* selects an external power meter that is used to measure the exact source power at the calibration point. Power meters (Pmtr1, Pmtr2, ...) must be configured explicitly in the System *Configuration – External Power Meters* dialog before they appear in the list.
- *Power Meter Config...* opens the *System Configuration – External Power Meters* dialog.
- *Perform Verification Sweep after Cal* enables or disables a verification sweep that the analyzer performs after the source power calibration. The sweep is also displayed in the calibration sweep diagram. Its purpose is to test the accuracy of the source power calibration after the final correction data has been acquired. The final power calibration results are applied starting with the first sweep after the verification sweep.

**Remote control:**
```
SOURce<Ch>:POWer<Pt>:CORRection:PMETer:ID
SOURce<Ch>:POWer<Pt>:CORRection:HAMonic[:ACQuire]
[SENSe<Ch>:]CORRection:POWer<Pt>:HARMonic:ACQuire
```

## Mixer

The *Mixer* submenu controls the measurement of the mixing products generated by an external mixer that is supplied with two independent RF signals. Together with the *Harmonic Distortion* measurement and the arbitrary configuration of generator and receiver frequencies *(Port Configuration),* the scalar mixer mode is included in option R&S ZVA-K4.

**Mixer Measurements**

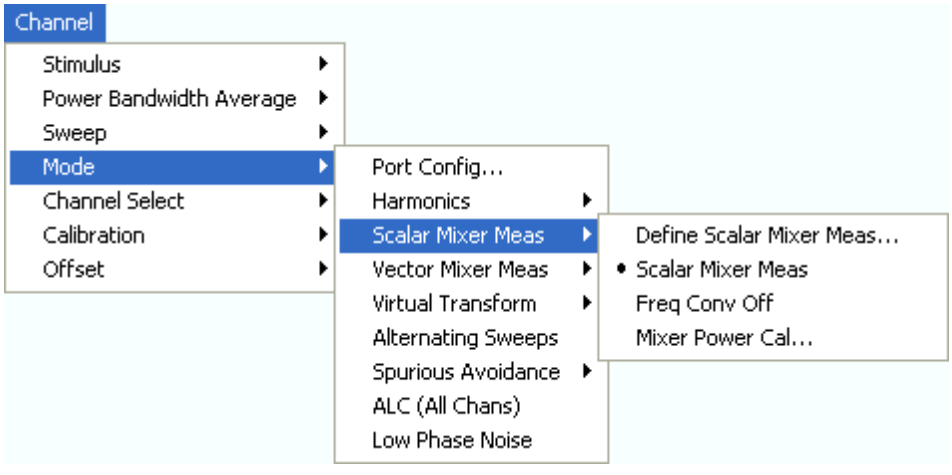
RF mixers convert an RF signal at one frequency into a signal at another frequency. The frequency that is to be shifted is applied at the RF input and the frequency shifting signal (from a local oscillator, LO) is applied to the RF mixer's LO port, resulting in an output signal at the mixer's Intermediate Frequency (IF) port.

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.

*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. In fact, transmission S-parameters obtained in a scalar mixer measurement only describe the magnitude ratios of the waves. The phase information, including the group delay, is meaningless.*

In the mixer mode the analyzer provides the following functionality:

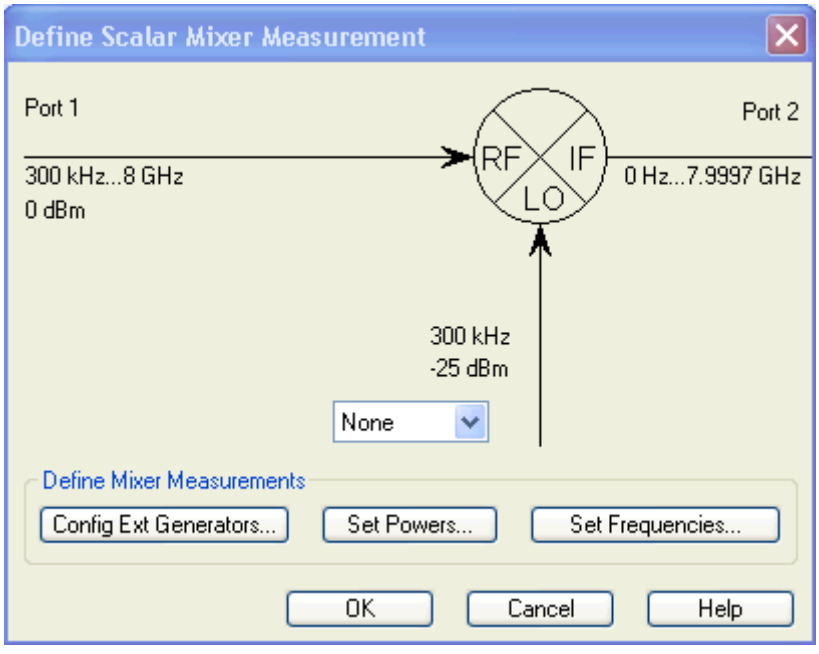
- Configuration of the RF and LO signals and measurement of the generated IF signal, see *Define Mixer Mode* and*Mixer Signal Diagrams*.
- Power calibration of the signal sources and of the IF receiver; see *Mixer Power Calibration*.
- The mixer mode can be used also to test important performance parameters of RF mixers such as frequency ranges, conversion loss, compression, and isolation.




- *Define Scalar Mixer Meas...* opens a dialog to configure the mixer input signals.
- *Scalar Mixer Meas* and *Freq Conv Off* toggle between the (frequency-converting) mixer mode and normal (non frequency converting) operation.
- *Mixer Power Cal...* opens a dialog to perform a power calibration of the signal sources and of the IF receiver.

## Define Scalar Mixer Meas...

Opens a configuration dialog for the mixer input signals.




In the upper part the *Define Mixer Measurement* dialog shows a diagram with the RF and LO signals, the mixing product (IF), and the current frequency and power ranges for all signals. Below the dialog provides the following control elements:

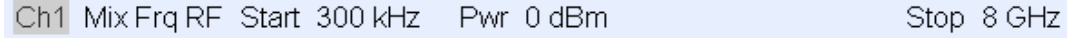
- The port selection list contains all analyzer ports or external generators which can provide the local oscillator (LO) signal. Ports no. 1 and 2 are always used for the RF signal and the measured IF signal (mixing product), so they are not included in the list. Generators must be configured explicitly in the *System Configuration – External Generators* dialog before they appear in the list. *None* means that the input signal at LO (if available) is not controlled by the analyzer.
- *Config Ext. Generators...* opens the *System Configuration – External Generators* dialog for adding and configuring external generators.
- *Set Powers...* opens a dialog to control the power of the RF and the LO signals.
- *Set Frequencies...* opens a dialog to control the frequency of the RF and the LO signal and the analyzer (IF signal) frequency.

**Remote control:** `[SENSe<Ch>:]FREQuency:CONVersion:MIXer:LOINternal <port_no>`
`[SENSe<Ch>:]FREQuency:CONVersion:MIXer:LOEXternal <gen_no>`

## Scalar Mixer Meas, Freq Conv Off

The softkey *Scalar Mixer Meas* activates the scalar mixer mode, where the analyzer ports are at different frequencies. *Freq Conv Off* disables all frequency-converting measurements, including e.g. an active harmonic distortion measurement.

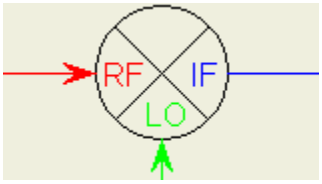
A *Mix Frq RF* label in the channel list indicates that the mixer mode is active.




**Remote control:** `[SENSe<Ch>:]FREQuency:CONVersion MIXer | FUNDamental`

## Mixer Signal Diagrams

The mixer signal diagrams show the parameters of the mixer input signals *(RF, LO)* and of the mixing product (IF signal, output). The diagrams appear in the *Mixer Measurement, Set Power,* and *Set Frequencies* dialogs.



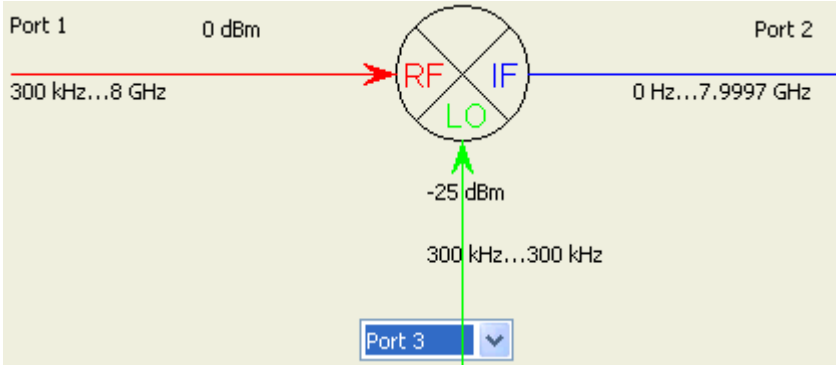

- The *RF* signal is the stimulus signal that the analyzer generates with the current channel settings. After a reset the frequency and power of the RF signal is as defined in the *Channel – Stimulus* menu. The RF signal parameters can be changed in the *Set Power...* and *Set Frequencies...* dialogs.
- 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.
- The *IF* signal is the mixer output signal (mixing product), which is at one of the following 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 *Set Frequencies...* dialog.

The signal description above with the swept RF signal and the LO signal at a fixed frequency corresponds to the default configuration. In the *Set Frequencies* dialog, you can select any of the signals as a *Sweep/CW* signal. You can set the frequency range of 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 diagram depends on the sweep type.

- **Frequency sweep**

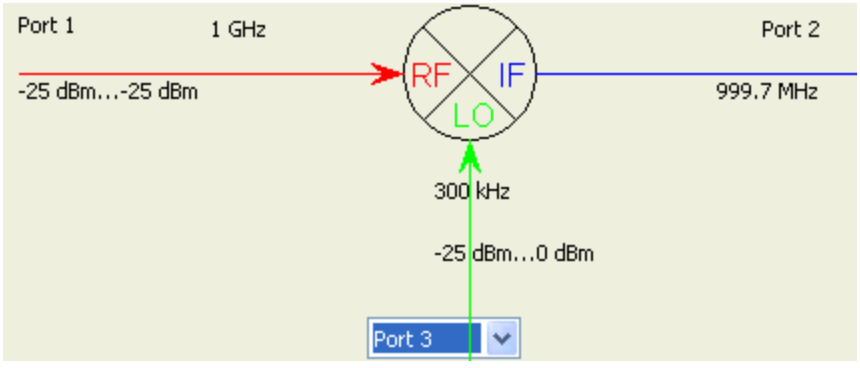
  For a frequency sweep, the signal diagram is labeled as follows:




  - *RF* signal: Fixed port number (1), CW power, frequency sweep range.
  - LO signal: Signal source (analyzer port or external generator), fixed power and frequency.
  - IF signal: Fixed port number (2), frequency range = (sweep range + LO) or |sweep range – LO|.

- **Power sweep**

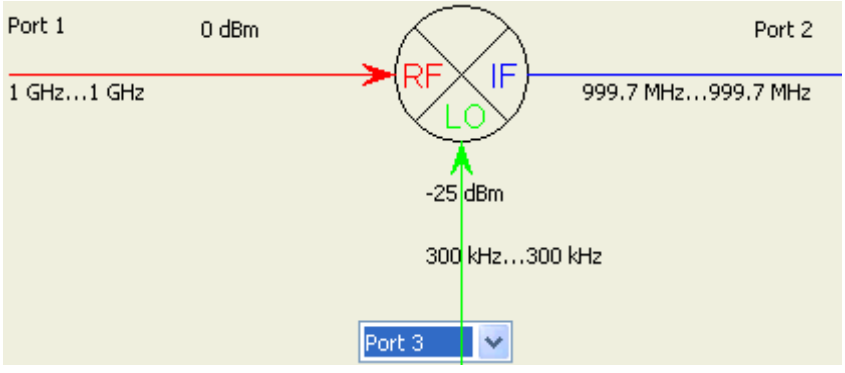
For a power sweep, the signal diagram is labeled as follows:




- *RF* signal: Fixed port number (1), power sweep range (or fixed power, if the LO signal is swept), CW frequency.
- LO signal: Signal source (analyzer port or external generator), fixed power (or power sweep range, if the RF signal is at a fixed power), CW frequency.
- IF signal: Fixed port number (2), fixed frequency = (RF + LO) or |RF – LO|.

- **Time or CW mode sweep**

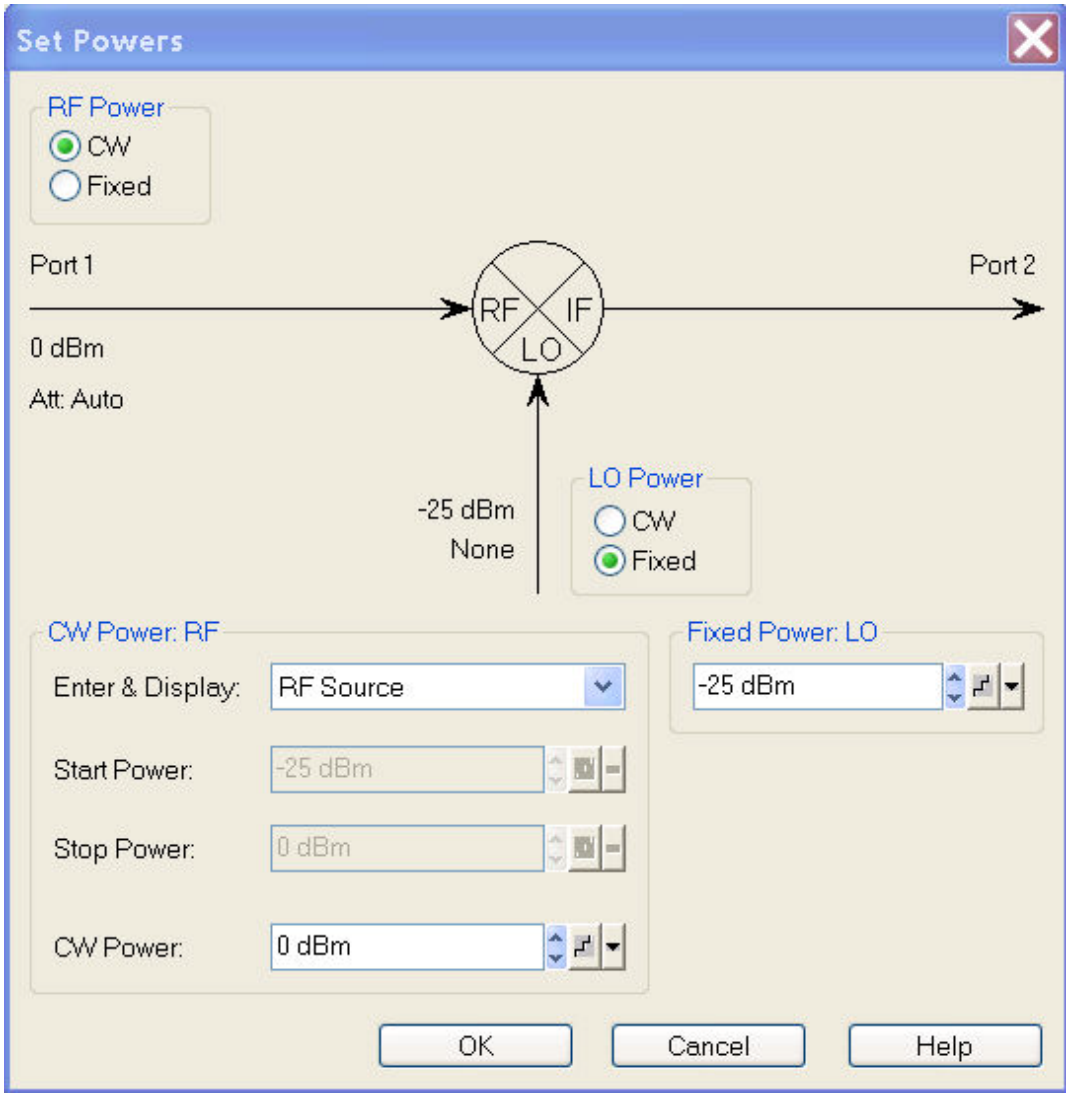
For a *Time* or *CW Mode* sweep, the signal diagram is labeled as follows:




- *RF* signal: Fixed port number (1), fixed power, CW frequency.
- LO signal: Signal source (analyzer port or external generator), fixed power, CW frequency.
- IF signal: Fixed port number (2), fixed frequency = (RF + LO) or |RF – LO|.

## Set Powers

The *Set Power* dialog defines the power of the RF and the LO signals.



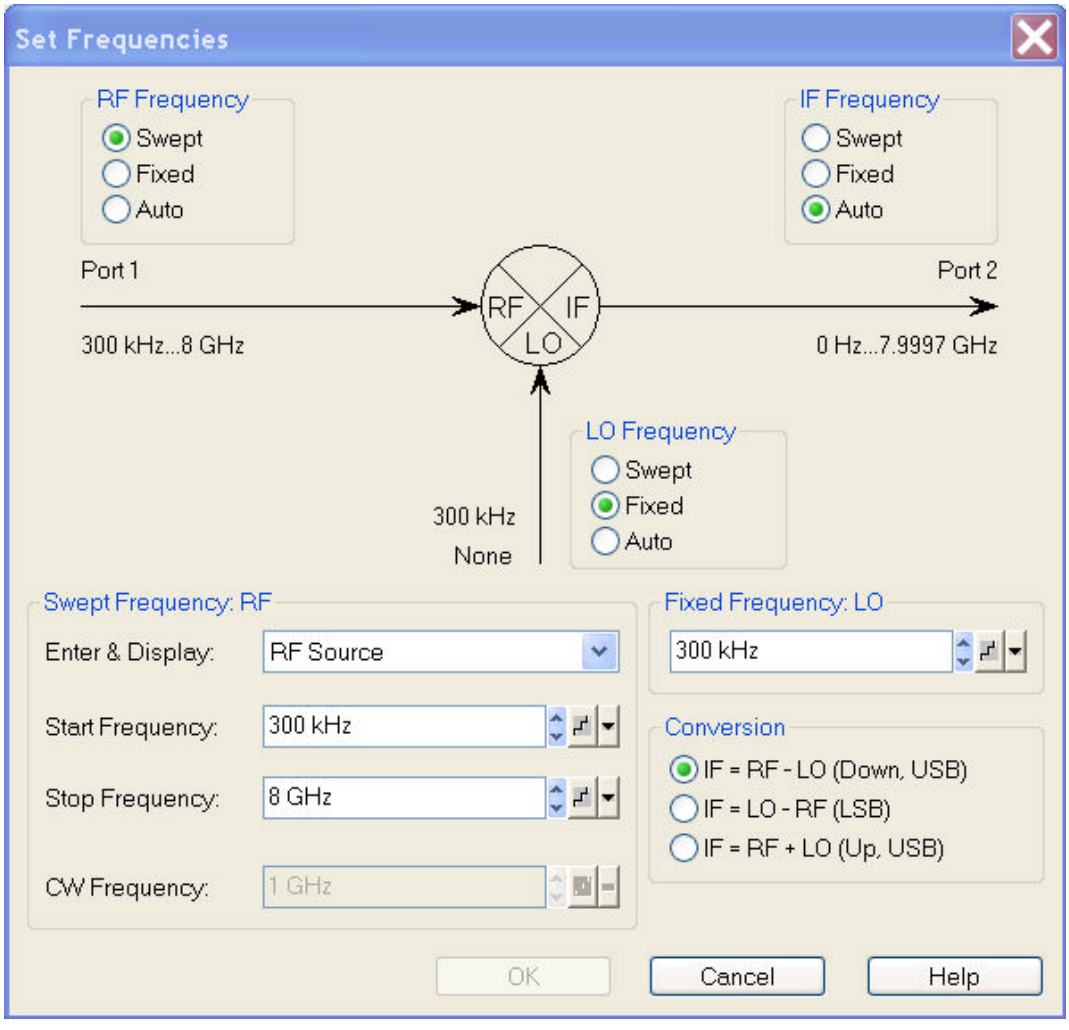

The diagram contains the mixer signal diagram and the following control elements:

- The *RF Power* or *LO Power* radio button groups specify how the power for each signal is defined. The roles of the mixer input signals RF and LO are interchangeable: One signal is at the analyzer's channel power *(Sweep* or *CW,* the signal is swept if a power sweep is active; it is at fixed power (CW) for the other sweep types), the other is at the *Fixed* power defined below.
- The input fields in the *Sweep* or *CW Power* panel (depending on the sweep type) define the channel power and overwrite the power settings in the *Channel – Stimulus* and *Channel – Power Bandwidth Average* menus. The sweep range or CW power is assigned to the signal with *Sweep/CW* power definition. *Enter & Display* can be used to select the *Swept* or *Auto* port for display and entry. Note: If you select the *Fixed* port, this port will be set to swept.
- *Fixed Power* defines the power to be assigned to the signal with *Fixed* power definition.

**Remote control:** `SOURce<Ch>:]FREQuency:CONVersion:MIXer:FUNDamental RF | LO`
`SOURce<Ch>:]FREQuency:CONVersion:MIXer:PFIXed <power>`

## Set Frequencies

Controls the frequency of the RF and the LO signal and the analyzer (IF signal) frequency.




The diagram contains the mixer signal diagram and the following control elements:

- The *RF/IF/LO Frequency* radio button groups specify how the frequency for each signal is defined. The roles of the three signals RF, IF, LO are interchangeable: One signal is at the analyzer's channel frequency *(Swept/CW,* the signal is swept if a frequency sweep type is active; it is a CW signal for the other sweep types), the second is at the *Fixed* frequency defined below, the frequency of the third signal is calculated according to the selected *Conversion* formula.
- The input fields in the *Swept Frequency* panel define the channel frequency and overwrite the *Channel – Stimulus* settings. The sweep range or CW frequency is assigned to the signal with *Swept/CW* frequency definition. *Enter & Display* can be used to select the *Swept* or *Auto* port for display and entry. Note: If you select the *Fixed* port, this port will be set to swept.
- *Fixed Frequency* defines the frequency to be assigned to the signal with *Fixed* frequency definition.
- *Conversion* selects the frequency of the signal with *Auto* frequency definition, see background information below.

**Conversion formulas**

The IF signal can be at the sum or at the difference of the RF and LO 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 perform measurements at different mixer output frequencies (e.g. to analyze the isolation for IF = RF or higher-order mixing products), use option R&S ZVA-K4, *Arbitrary Generator and Receiver Frequencies*.

In a frequency sweep mode where one of the input frequencies varies over a definite range, the IF signal also covers a frequency range of the same width. An error message is displayed if the IF frequency exceeds the maximum frequency range of the analyzer.

If the RF or LO signal is in *Auto* mode, the formulas for IF are automatically converted, and the analyzer calculates the RF or LO frequency from the other frequencies.

**Remote control:**

```
[SENSe<Ch>:]FREQuency:CONVersion MIXer:FFIXed <frequency>
[SENSe<Ch>:]FREQuency:CONVersion MIXer:FIXed RF | IF | LO
[SENSe<Ch>:]FREQuency:CONVersion MIXer:FUNDamental RF | IF | LO
[SENSe<Ch>:]FREQuency:CONVersion MIXer:TFRequency DCLower |
DCUPper | UCONversion
[SENSe<Ch>:]FREQuency:STARt etc.
```

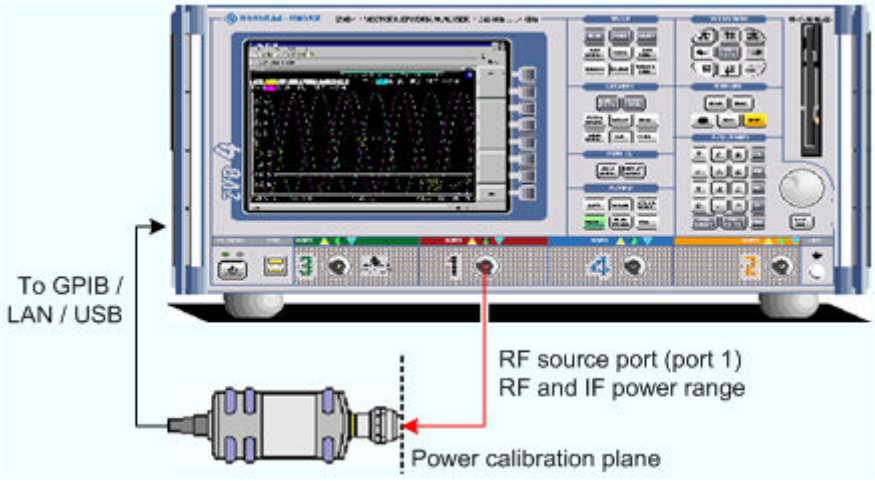
## Mixer Power Cal...

Opens a dialog to perform a power calibration of the signal sources and of the IF receiver. A power calibration ensures accurate levels of the mixer input signals (RF and LO source power calibration) and an accurate power measurement of the mixer output signal (IF receiver calibration). The necessary calibration steps are automatically performed across the entire frequency range of the active mixer measurement *(Set Frequencies).*

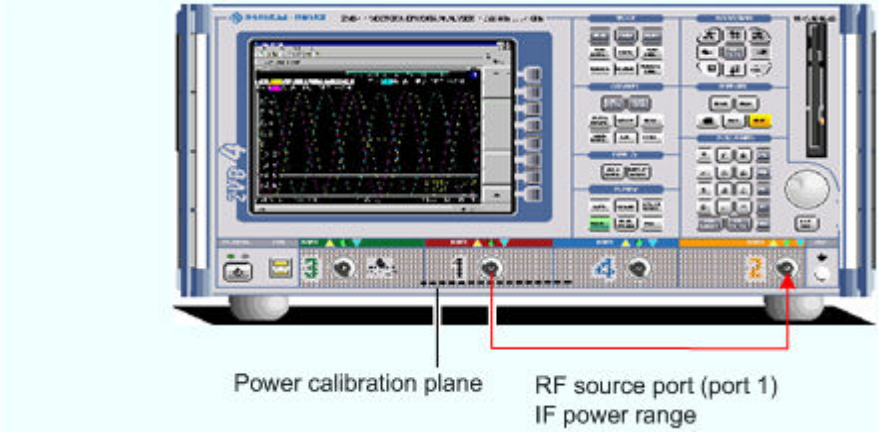
**Mixer Power Cal procedure and example**

Due to the different frequency ranges of the input and output signals and the external equipment needed, the mixer power calibration must be performed in several steps. For a test setup where the RF and LO signals are provided by port 1 and an external generator, respectively, where the RF signal is swept, and where the IF signal is analyzed at port 2, the following test setups are required:

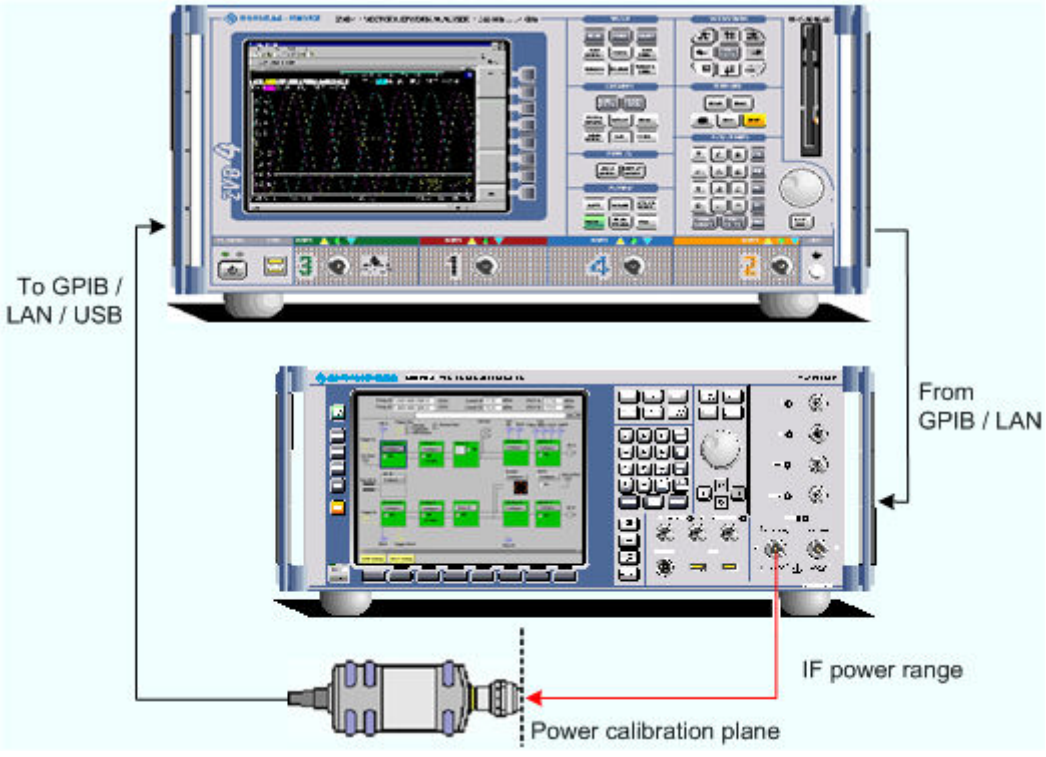
11. Source power calibration for the RF signal. A power sensor is connected to port 1. The RF power (red) is calibrated over the selected RF sweep range. As a preparative for step 2 the analyzer also calibrates the source at the fixed IF frequency. The correction data for both calibration sweeps is stored.

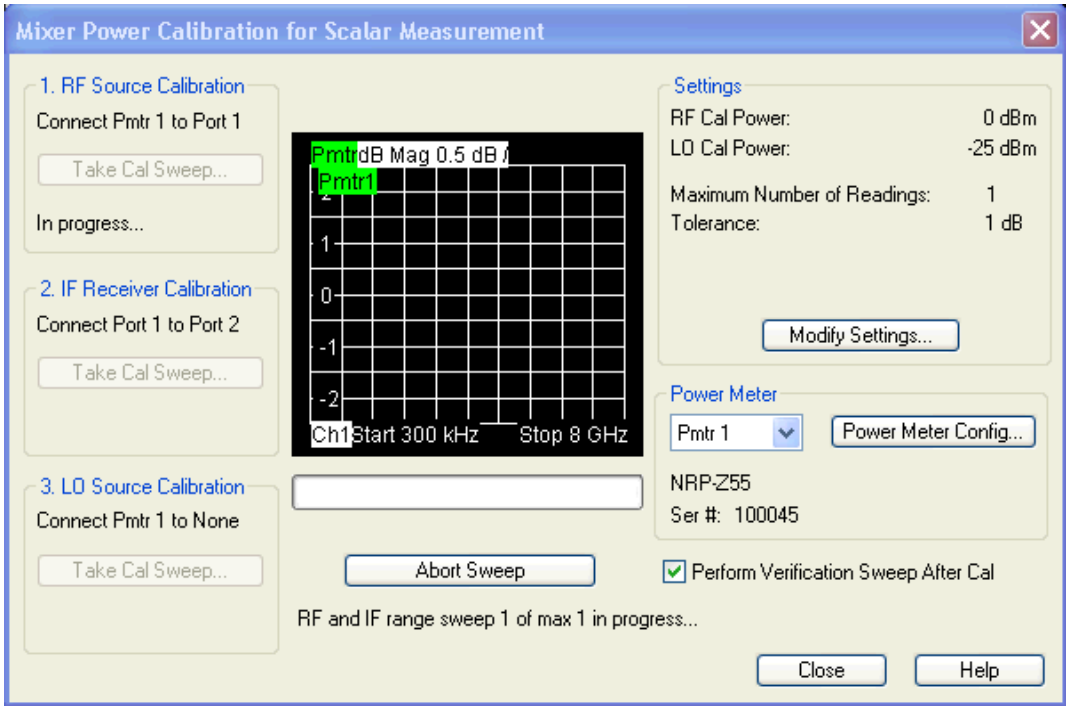



12. Receiver power calibration for the IF signal. No external device is needed. The receiver (port 2) is calibrated at the IF frequency using the source signal from port 1 (red) calibrated in the first step.




13. Source power calibration for the LO signal, provided by an external generator. A power sensor is connected to the generator's RF output (red signal). The analyzer controls the generator, its RF power (red) is calibrated at the selected IF frequency. No RF connection to or from the network analyzer is needed.




- *Take Cal Sweep* in the three numbered panels start the calibration sweeps for the RF source, IF receiver, and the LO source, respectively. The measured power is shown in the diagram in the center of the dialog and the sweep progress in the progress bar below.
- *Abort Sweep* stops the current sweep, e.g. in order to change the sweep settings.
- The calibration sweeps are performed according to the settings shown in the *Settings* panel in the upper right section of the dialog. *Modify Settings* opens the *Modify Source Power Cal Settings* dialog to modify the essential settings of the next calibration sweep.
- *Power Meter Config...* opens the *System Configuration – External Power Meters* dialog for

adding and configuring external power meters.

- *Perform Verification Sweep after Cal* enables or disables a verification sweep that the analyzer performs after each source power calibration. The sweep is also displayed in the calibration sweep diagram. Its purpose is to test the accuracy of the source power calibration after the final correction data has been acquired.

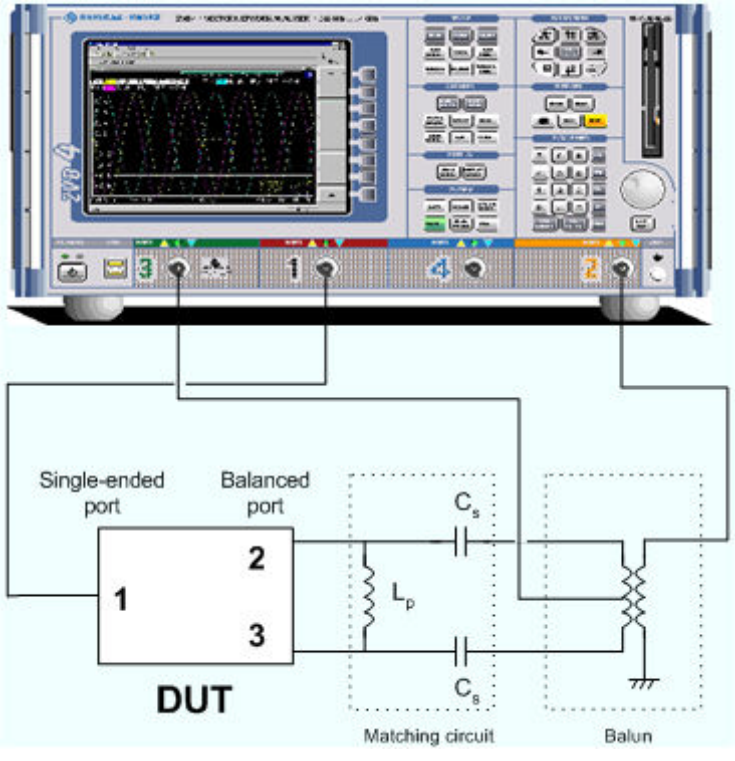
**Remote control:**
```
SOURce<Ch>:POWer<Pt>:CORRection:PMETer:ID
SOURce<Ch>:POWer<Pt>:CORRection:MIXer:RF[:ACQuire]
[SENSe<Ch>:]CORRection:POWer<port_no>:MIXer:IF:ACQuire
SOURce<Ch>:POWer<Pt>:CORRection:MIXer:LO[:ACQuire]
```

## Virtual Transform

The *Virtual Transform* submenu defines virtual networks to be added to/removed from the measurement circuit for a DUT with single ended or balanced ports. The submenu is available for analyzer models with arbitrary numbers of ports.

**Embedding a DUT into a matching network**

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) in order 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 a number of advantages:

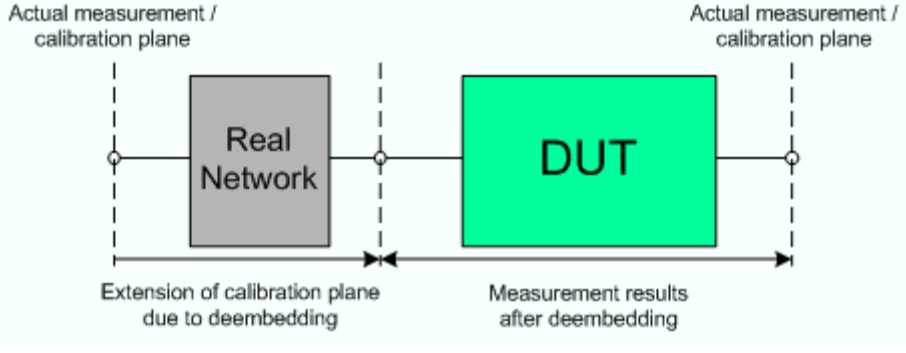
- The measurement uncertainty is not impaired by the tolerances of real test fixtures.
- There is no need to fabricate test fixtures with integrated matching circuits for each type of DUT.
- Calibration can be performed at the DUT's ports. If necessary (e.g. for compensating for the effect of a test fixture) it is possible to shift the calibration plane using length offset parameters.

**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 in order 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.

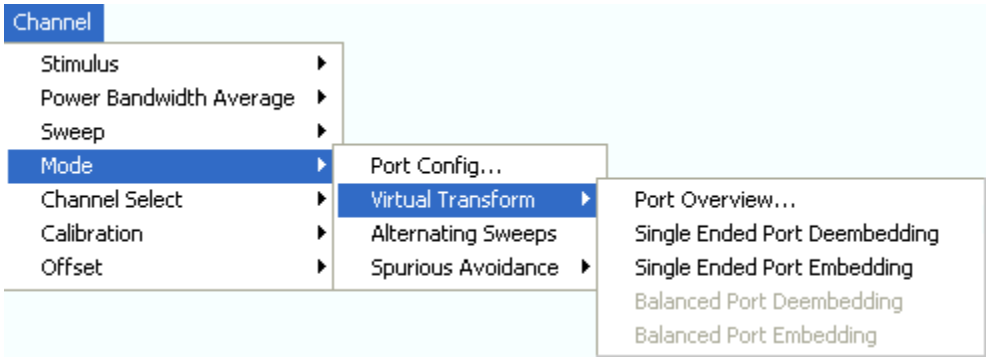
To be numerically removed, the real network must be described by a set of S-parameters or by an equivalent circuit of lumped elements. Deembedding the DUT effectively extends the calibration plane towards the DUT ports, enabling a realistic evaluation of the DUT without the distorting network. Deembedding can be combined with length offset parameters.

The simplest case of one-port deembedding can be depicted as follows:




The embedding/deembedding function in the *Virtual Transform* menu has the following characteristics:

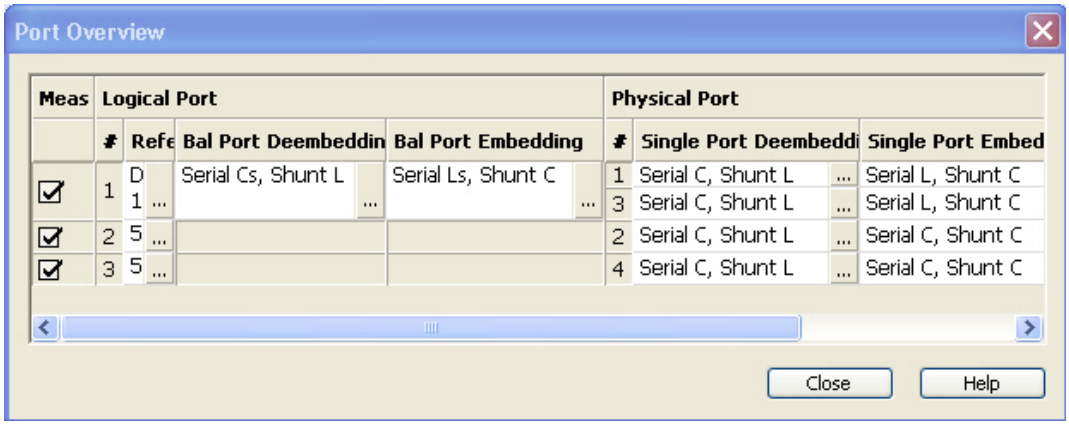
- Embedding and deembedding can be combined with balanced port conversion: the (de-)embedding function is available for single ended and balanced ports.
- A combination of four-port and two-port networks (not necessarily both) can be applied to balanced ports; two-port networks can be applied to single ended ports.
- 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.




- *Port Overview...* opens a dialog providing an overview of all analyzer ports with their reference impedances and transformation networks.
- *Single Ended Port Deembedding/Embedding* directly opens the configuration dialog for the single ended circuits and activates or deactivates (de-)embedding.
- *Balanced Port Deembedding/Embedding* directly opens the configuration dialog for the balanced circuits and activates or deactivates (de-)embedding. These commands are enabled only after a balanced port configuration has been defined.

## Port Overview

Provides an overview of analyzer ports with their reference impedances and transformation networks for (de-)embedding.

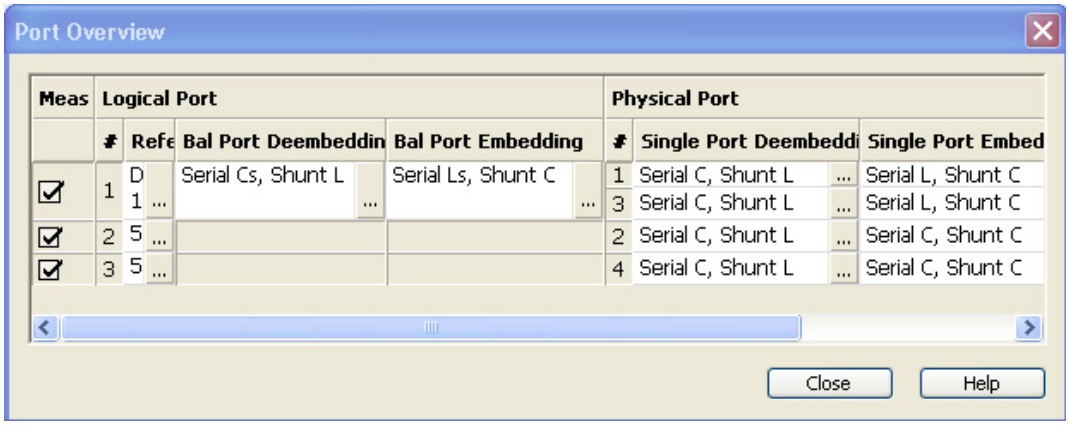



Each row in the *Port Overview* table corresponds to one logical or physical analyzer port; the number of rows in the different columns is equal to the number of balanced ports or physical test ports of the analyzer. The ... buttons open different dialogs for test port configuration:

- *Reference Impedance* defines complex impedances for balanced test ports.
- *Bal Port Deembedding* and *Bal Port Embedding* show the 4-port transformation networks to be added or removed at balanced (logical) ports. *Off* denotes that no (de)embedding operation is performed.
- *Single Port Deembedding* and *Single Port Embedding* show the 2-port transformation networks to be added or removed at single ended (physical) or balanced (logical) ports. *Off* denotes that no (de)embedding operation is performed.

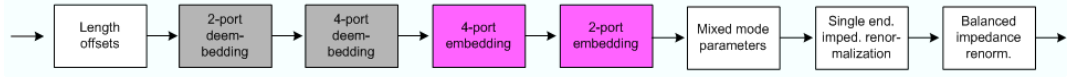
## Combination of 2-Port and 4-Port (De-)Embedding Networks

In the *Port Overview* dialog, it is possible to select a combination of 4-port and 2-port networks for (de-)embedding at balanced (logical) ports, and to select 2-port networks for (de-)embedding at single ended (physical) ports. The following example shows the full configuration for a four-port analyzer where the physical ports 1 and 3 form a balanced port with the logical port no. 1 and the remaining ports 2 and 4 are single ended.



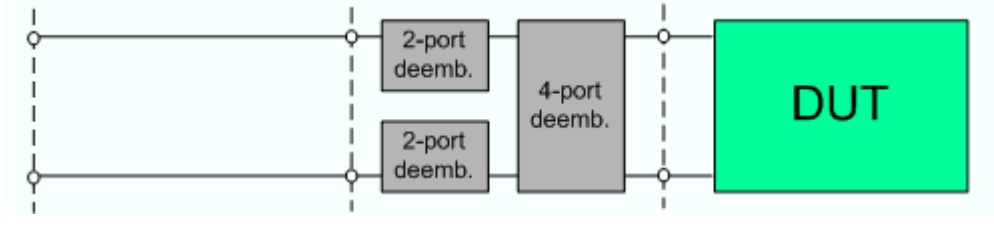

The port overview configuration above corresponds to the measurement circuits shown in steps 1 to 5 below.

The different steps for deembedding and embedding are carried out in the following order:

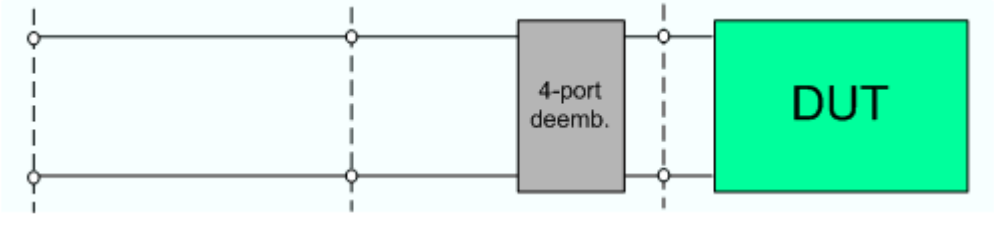



This means that the real networks are removed before virtual networks are added. For a a single balanced port with all (de-)embedding networks enabled, the 4 (de-)embedding steps are carried out in the following order:

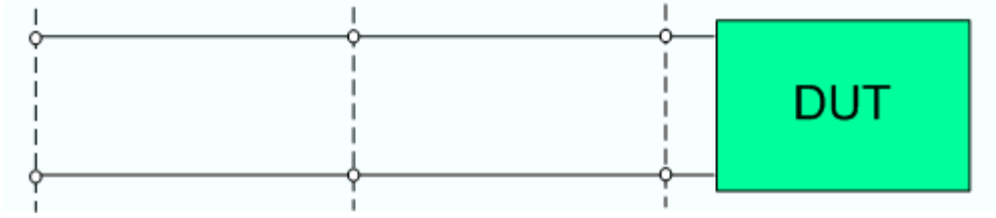
14. **Initial situation: DUT embedded in 2-port and 4-port networks (only 1 port shown)**



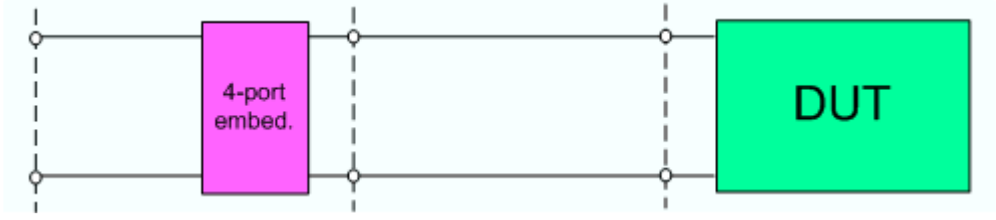

15. **2-port deembedding**



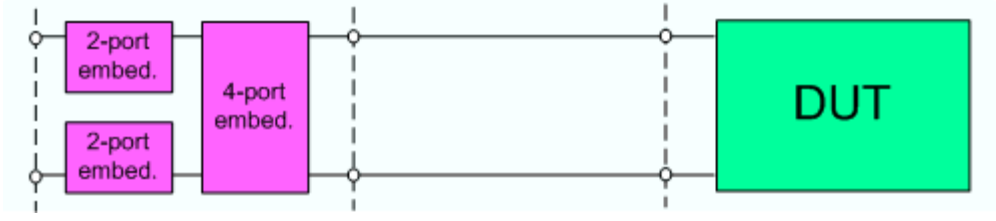

16. **4-port deembedding**




17. **4-port embedding**




18. **2-port embedding**

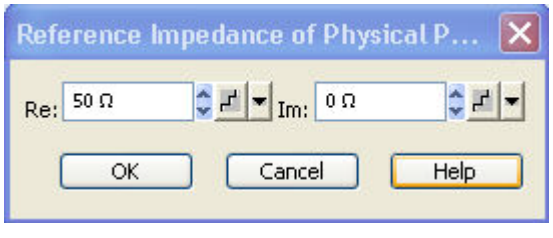



## Reference Impedances...

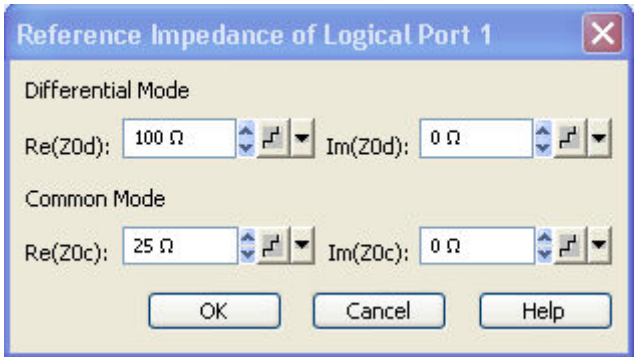
Changes the reference impedances of the analyzer ports. This is often referred to as renormalization of port impedances. Renormalization means that the measurement results measured at 50 Ω (75 Ω) are converted into results at arbitrary port impedance.

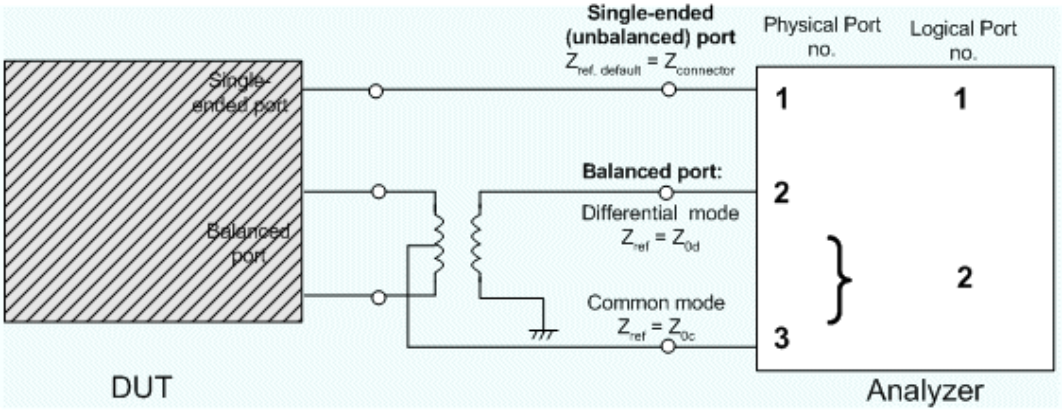
- Renormalization of the physical port impedances affects e.g. S-parameters and wave quantities in *Power* representation.
- Renormalization of the balanced port impedances affects all measured quantities *(Trace – Measure)* that the analyzer provides for balanced ports.

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

**Conversion formula for 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 S-matrix $S_1$ can be calculated from $S_0$ and the reference impedances $Z_{0i}$, $Z_{1i}$.according to

$$S_1 = Q^{-1} \cdot (E - S_0 G) \cdot (S_0 - G) \cdot Q,$$

with the unit matrix E and the additional matrices

$$Q = \begin{pmatrix} \sqrt{\frac{\mathrm{Re}(Z_{01})}{\mathrm{Re}(Z_{11})}}\left|\frac{Z_{11}}{Z_{01}}\right|\frac{Z_{11}+Z_{01}}{2Z_{11}} & 0 & \dots & 0 \\ 0 & \sqrt{\frac{\mathrm{Re}(Z_{02})}{\mathrm{Re}(Z_{12})}}\left|\frac{Z_{12}}{Z_{02}}\right|\frac{Z_{12}+Z_{02}}{2Z_{12}} & \dots & 0 \\ \dots & \dots & \dots & 0 \\ 0 & 0 & \dots & \sqrt{\frac{\mathrm{Re}(Z_{0n})}{\mathrm{Re}(Z_{1n})}}\left|\frac{Z_{1n}}{Z_{0n}}\right|\frac{Z_{1n}+Z_{0n}}{2Z_{1n}} \end{pmatrix},$$

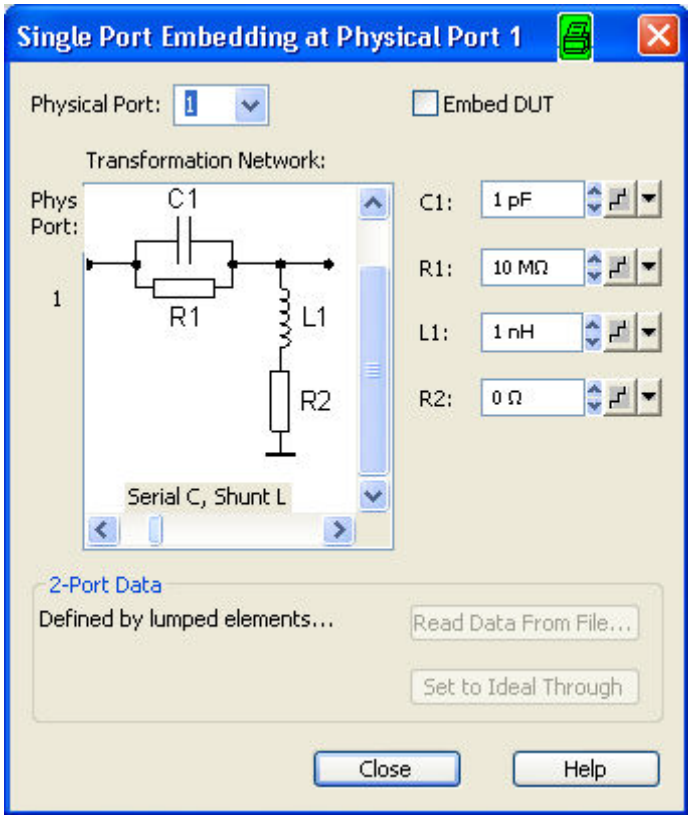
$$G = Q \cdot Q^{-1} = \begin{pmatrix} \frac{Z_{11}-Z_{01}}{Z_{11}+Z_{01}} & 0 & \dots & 0 \\ 0 & \frac{Z_{12}-Z_{02}}{Z_{12}+Z_{02}} & \dots & 0 \\ \dots & \dots & \dots & 0 \\ 0 & 0 & \dots & \frac{Z_{1n}-Z_{0n}}{Z_{1n}+Z_{0n}} \end{pmatrix}.$$

In true differential mode, a renormalization formula for the balanced wave quantities is needed; refer to section Wave Quantities and Ratios.

**Remote control:**
```
SENSe:PORT<phys_port>:ZREFerence <real> [,<imaginary]
SENSe:LPORT<log_port>:ZCOMmon <real> [,<imaginary]
SENSe:LPORT<log_port>:ZDIFferent <real> [,<imaginary]
```

## Single Port Embedding/Deembedding

Selects a 2-port transformation network for single port (de-)embedding, defines its parameters, assigns it to a physical port and enables (de-)embedding. Single port (de-)embedding can be used for balanced (logical) as well as for single ended (physical) ports. The two dialogs for deembedding and embedding are identical except for their inverse effect.




The dialogs contain the following control elements:

- *Physical Port* is the physical analyzer port for the added or removed circuit. The transformation networks are defined such that the physical analyzer test port is connected to the left of the circuit; the DUT port is on the right side.
- *(De-)embed DUT* enables or disables the (de-)embedding function.
- *Transformation Network* contains all available 2-port networks (see below). Networks are either defined by lumped elements or by means of imported S-parameter data. The element parameters (C, R, L) for the selected network are displayed on the right side.
- *Read Data From File...* is enabled as long as the *2-Port Data* network is active. This network is defined by its S-parameters stored in a two-port Touchstone file (*.s2p). No additional parameters are required.
- *Set to Ideal Through* is enabled as long as the *2-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.

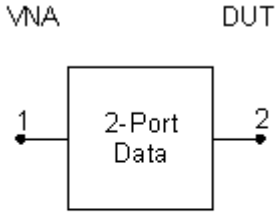
**Circuit models for 2-port networks**

The lumped element 2-port transformation networks for (de-)embedding consist of the following two basic circuit blocks:

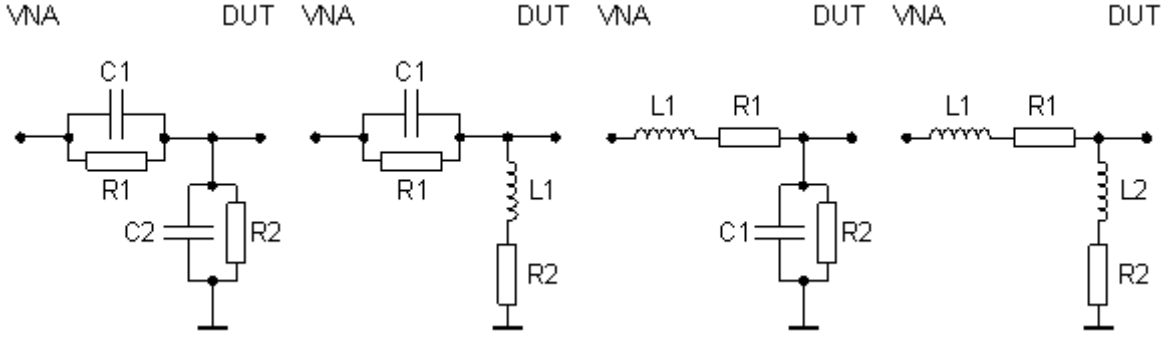
- A capacitor C connected in parallel with a resistor.
- An inductor L connected in series with a resistor.

The 2-port transformation networks comprise all possible combinations of 2 basic blocks, where one block represents a serial, the other 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.

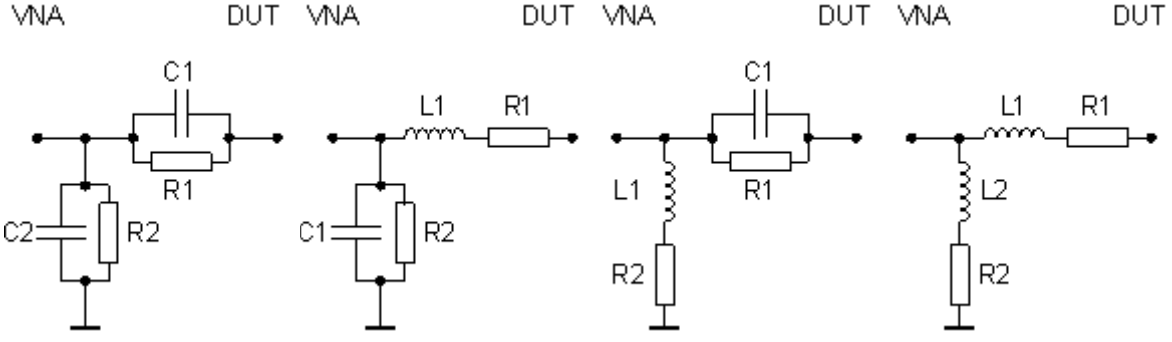
The first network is defined by its S-parameters stored in an imported two-port Touchstone file (*.s2p). No additional parameters are required.




The following networks are composed of a serial C or L (as seen from the test port), followed by a shunt C or L. They are named *Serial C, Shunt C / Serial C, Shunt L / Serial L, 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 C / Shunt C, Serial L / Shunt L, Serial C / Shunt L, Serial L.*



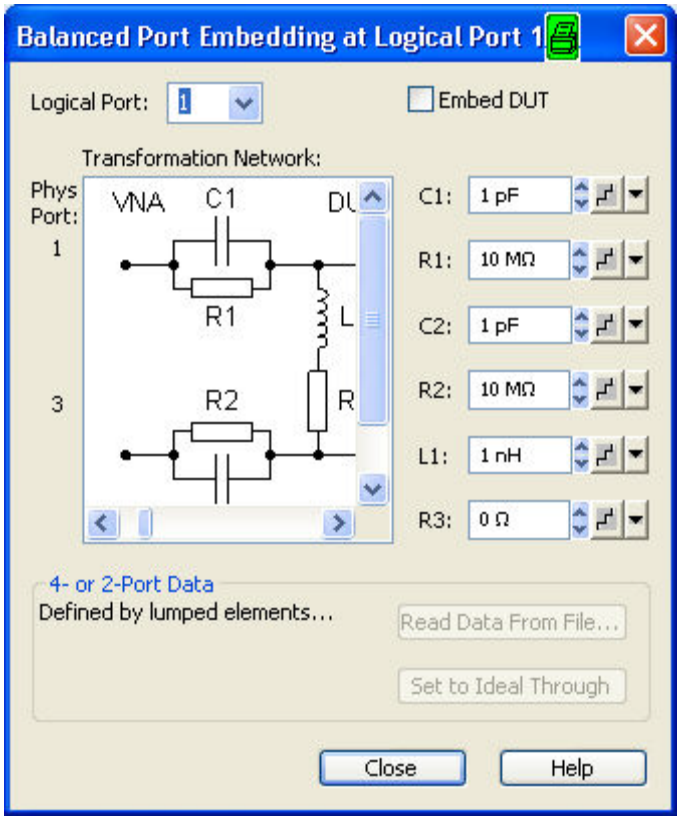

**Remote control:**

```
CALCulate<Ch>:TRANsform:VNETwork:SENDed...
MMEMory:LOAD:VNETworks<Ch>:SENDed:DEEMbedding<Ph_pt>
'<file_name>'
MMEMory:LOAD:VNETworks<Ch>:SENDed:EMBedding<Ph_pt>
'<file_name>'
```

## Balanced Port Embedding/Deembedding

Selects a 4-port transformation network for balanced port (de-)embedding, defines its parameters, assigns it to a physical port and enables (de-)embedding. The two dialogs for deembedding and

embedding are identical except for their inverse effect. The dialogs are available only after a balanced port configuration has been defined.




The dialogs contain the following control elements:

- *Physical Port* is the physical analyzer port for the added or removed circuit. The transformation networks are defined such that the physical analyzer test port is connected to the left of the circuit; the DUT port is on the right side.
- *(De-)embed DUT* enables or disables the (de-)embedding function.
- *Transformation Network* contains all available 4-port networks (see below). Networks are defined by lumped elements or by means of imported S-parameter data. The element parameters (C, R, L) for the selected network are displayed on the right side.
- *Read Data From File...* is enabled as long as one of the networks involving two-port or four-port S-parameter data is active. The S-parameters are read from two-port (*.s2p) or four-port (*.s4p) Touchstone files. Each two-port file can be assigned to one of two different physical ports, to be selected by means of the *Phys. Port* radio buttons.
- *Set to Ideal Through* is enabled as long as one of the networks involving two-port or four-port S-parameter data is active. An imported S-parameter set is replaced by the S-parameters of an ideal through connection, which eliminates the transformation network.

**Circuit models for 4-port networks**

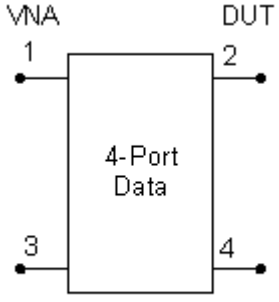
The lumped element 4-port transformation networks for (de-)embedding consist of the following two basic circuit blocks:

- A capacitor C connected in parallel with a resistor.
- An inductor L connected in series with a resistor.

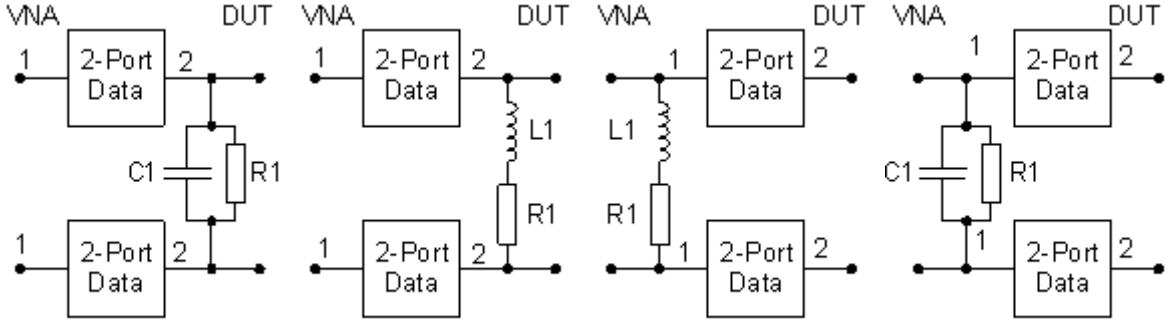
The transformation networks comprise various combinations of 3 basic circuit blocks, where two blocks represent serial elements, the third a shunt element. In the default setting the resistors are not effective,

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.

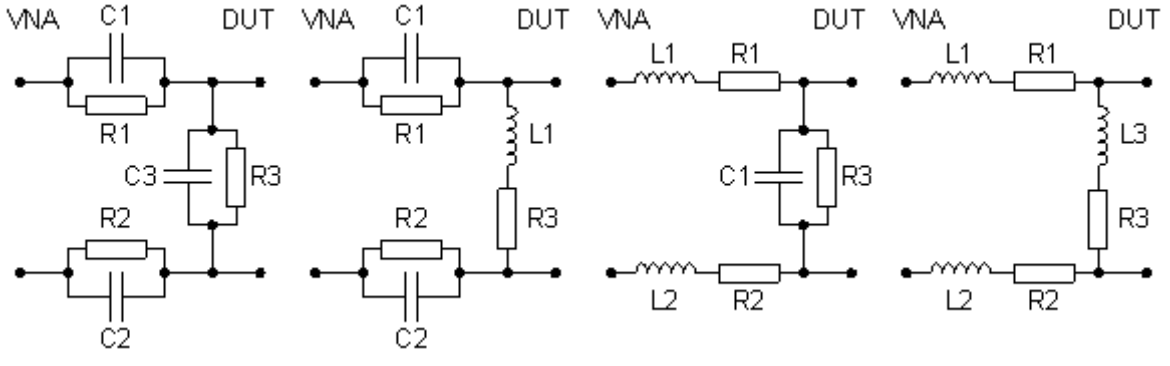
The first network is defined by its S-parameters stored in an imported four-port Touchstone file (*.s4p). No additional parameters are required.



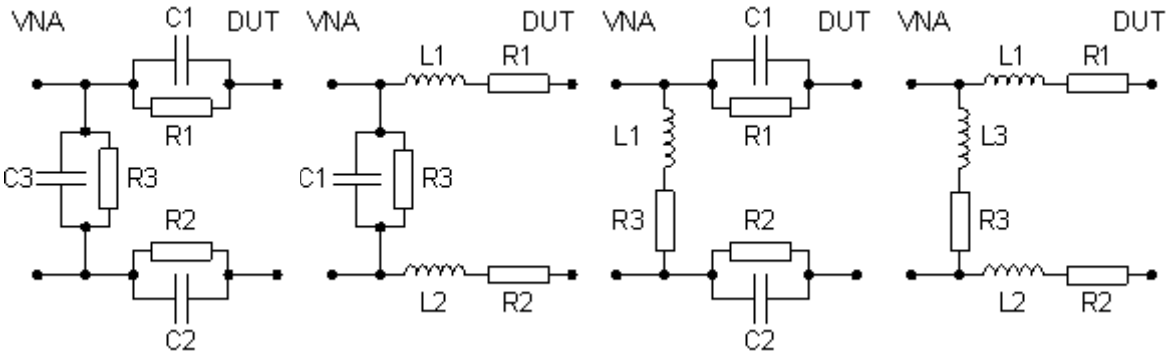

The following networks are composed of a shunt C or L and two serial elements, described by means of imported 2-port S-parameters. They are named *Serial 2-port, Shunt C / Serial 2-port, Shunt L / Shunt L, Serial 2-port / Shunt C, Serial 2-port.*




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 C / Serial Cs, Shunt L / Serial Ls, 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 C, Serial Cs / Shunt C, Serial Ls / Shunt L, Serial Cs / Shunt L, Serial Ls.*




**Remote control:**

```
CALCulate<Ch>:TRANsform:VNETwork:BALanced...
MMEMory:LOAD:VNETworks<Ch>:BALanced:DEEMbedding<Log_pt>
'<file_name>', PMAIn, PSECondary
MMEMory:LOAD:VNETworks<Ch>:BALanced:EMBedding<Log_pt>
'<file_name>', PMAIn, PSECondary
```

## Alternate

Activates or deactivates the operating mode where the analyzer performs only one partial measurement per sweep.

- In normal sweep mode *(Alternate* disabled), 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.
- *Alternate* reverses the order of partial measurements and sweeps: A partial measurement is performed at all sweep points (partial sweep) before the hardware settings are changed and the next partial measurement is carried out in an additional sweep.

The *Alternate* mode is also valid during a system error correction.

**Advantages of both sweep modes, alternative settings**

If the settling time between adjacent frequency points is larger than the settling time between the partial measurements, then the *Alternate* measurementis faster than a normal sweep so that a smaller sweep times can be set. On the other hand, an *Alternate* measurement provides a result only during the last partial sweep.

Use the *Alternate* 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, deactivate the *Alternate* mode.

As an alternative to activating the *Alternate* mode, it is possible to insert a *Meas. Delay* before each partial measurement and thus improve the accuracy. The delay slows down the measurement.

**Relation to trigger settings**

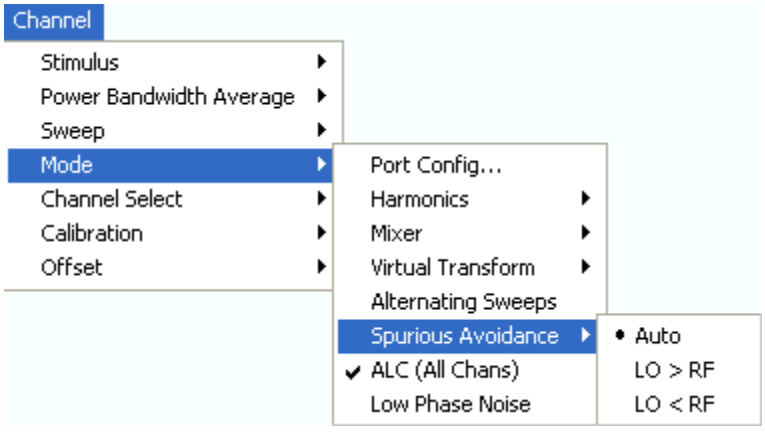
In triggered measurements, *Alternate* has no effect if the triggered measurement sequence is identical to a single sweep point. The following table shows how the analyzer performs a sweep comprising m sweep points, assuming that each of them requires n partial measurements.

| Triggered Meas. Sequence | Alternate On | Alternate Off |
|---|---|---|
| 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 |

**Remote control:** `[SENSe<Ch>:]COUPle ALL | NONE`

## Spurious Avoidance

The *Spurious Avoidance* submenu defines whether the analyzer measures with a local oscillator frequency LO below or above the RF input frequency.



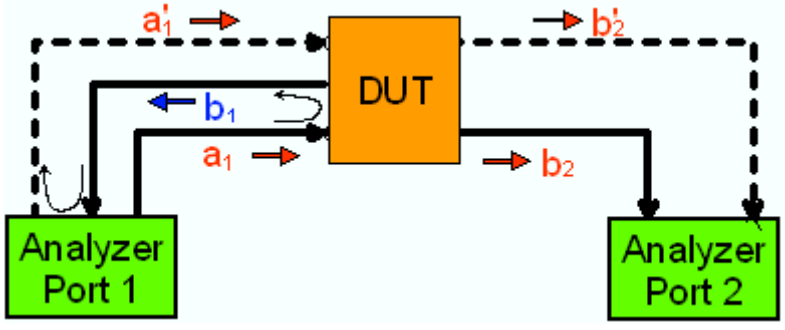

This feature can be used to eliminate known spurious components in the input signal that can distort the measurement, especially in the low frequency range.

- In *Auto* mode, the analyzer auto-selects the local oscillator frequency, depending on the receiver (RF) frequency and the test port. This mode systematically avoids known spurious signals provided that no frequency conversion occurs in the test setup.
- *LO > RF* means that the LO frequency is always above the measured RF frequency. This mode is appropriate for avoiding single, known spurious signals.
- *LO > RF* means that the LO frequency is always below the measured RF frequency. This mode is appropriate for avoiding single, known spurious signals.

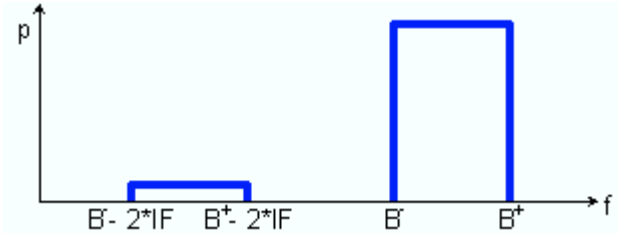
In the presence of several spurious signals, setting the *Spurious Avoidance* parameter globally may not be sufficient. To improve the result, perform a *Segmented Frequency* sweep and assign independent LO frequencies to the individual sweep segments.

**Application example**

Consider the following test setup with strongly reflecting DUT (e.g. a bandpass in its stop band) that is measured in transmission. $a_1$ is generated at a frequency RF.The reflected wave $b_1$ falls into the receiver mixer of the analyzer port 1, where 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 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 control:** `[SENSe<Ch>:]FREQuency:SBANd POSitive | NEGative | AUTO`

## ALC (All Chans)

Enable or disables the Automatic Level Control (ALC) for all channels. ALC keeps the level of the a waves (source level) at a constant value, irrespective of the DUT's input impedance. The measurement speed is slightly reduced.

*If the analyzer cannot reach the designated source level at a particular sweep point, it displays the message Ports n: Source level failure. Check the data sheet for an overview of the available source levels.*

**Remote control:** `DIAGnostic:ALC:SETTings[:STATe] ON | OFF`

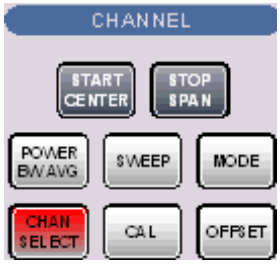
### Low Phase Noise

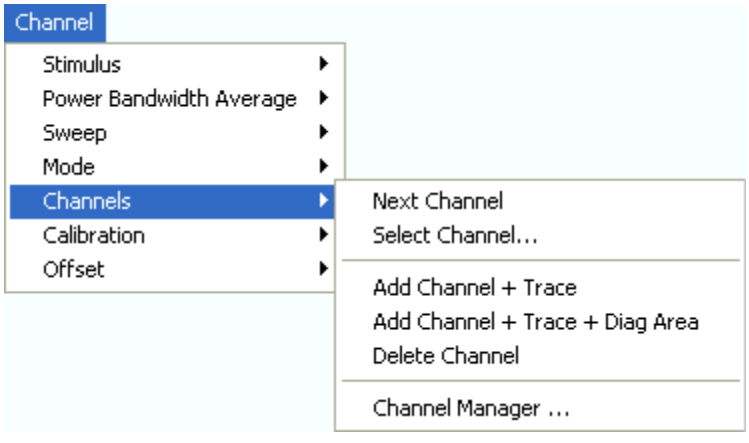
Reduces the phase noise of the source signals. The measurement speed is slightly reduced.

**Remote control:** `[SENSe<Ch>:]FREQuency<1>:LPNoise ON | OFF`

## Channel Select

The *Channel Select* submenu provides functions to create and delete channels and select a channel as the active channel.

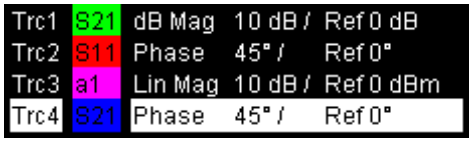



- *Next Channel* selects the next channel as the active channel (disabled if only one channel is defined).
- *Select Channel* opens a box to select an arbitrary channel of the active setup as the active channel (disabled if only one channel is defined).
- *Add Channel + Trace* creates a new channel and a new trace in the active diagram area.
- *Add Channel + Trace + Diag Area* creates a new channel and a new trace in a new diagram area.
- *Delete Channel* deletes the active channel.
- *Channel Manager* opens a dialog to perform the previous actions systematically, rename channels.

**Active and inactive traces and channels**

A window can display several diagram areas 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 area (Trc 4 in the figure below):




A mouse click onto a trace in the list selects the trace as the active trace. Alternatively, use the functions of the *Trace – Traces* menu.

The active channel is the channel belonging to the active trace. The channels of all traces in a diagram area 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 (Ch1 in the example below, with two associated traces).

A mouse click onto a trace in the trace list selects the channel associated to the trace as the active channel. Channels with no traces are not indicated in the diagram areas but can be accessed via the *Channel Manager*.

You can monitor the channel activity using the `OUTPut<Ch>:UPORt[:VALue] <numeric_value>` command and the output signals at pins 8 to 11 of the USER CONTROL connector.

## Next Channel

Selects the next channel in a list of defined channels as the active channel. This function is disabled if the current setup contains only one channel.

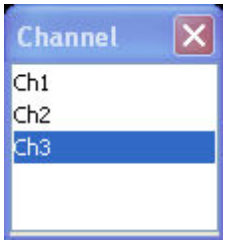
If one or several traces are assigned to the next channel, one of these traces becomes the active trace.

The order of all channels belonging to a setup is given by the channels' creation time. By default, the channels are named Ch1, Ch2, ... so that Ch<n> follows Ch<n – 1>. This order is always maintained, even if channels are renamed, invisible (because no traces are assigned to them) or distributed over several diagram areas.

**Remote control:** The numeric suffix `<Ch>` appended to the first-level mnemonic of a command selects a channel as active channel.

## Select Channel

Opens a box to select an arbitrary trace of the active setup as the active trace. This function is disabled if the current setup 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 belonging to a setup 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.

**Remote control:** The numeric suffix `<Ch>` appended to the first-level mnemonic of a command selects a channel as active channel.

## Add Channel + Trace

Creates a new channel and a new trace, which is displayed in the active diagram area. The new channel settings (including a possible channel calibration) are identical to the previous channel settings; the trace is created with the trace settings of the former active trace but displayed with another color. The former and the new active trace are superimposed but can be easily separated, e.g. by changing the *Reference Position.*

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.

To create a new trace in the active channel, use the *Trace – Traces – Add Trace* function. To create a new channel and a new trace and display it in a new diagram area, use *Add Channel + Trace + Diag. Area*.

**Remote control:**
```
CONFigure:CHANnel<Ch>[:STATe] ON
CALCulate<Ch>:PARameter:SDEFine "<Trc_name>", "<Parameter>"
DISPlay:WINDow<No>:TRACe:FEED "<Trc_name>"
```

## Add Channel + Trace + Diag. Area

Creates a new channel and a new trace, which is displayed in a new diagram area. The new channel settings (including a possible channel calibration) are identical to the previous channel settings; the trace is created with the trace settings of the former active trace but displayed with another color.

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.

To create a new trace in the active channel, use the *Trace – Traces – Add Trace* function. To create a new channel and a new trace and display it in the active diagram area, use *Add Channel + Trace.*

**Remote control:**
```
CONFigure:CHANnel<Ch>[:STATe] ON
```

```
CALCulate<Ch>:PARameter:SDEFine "<Trc_name>", "<Parameter>"
DISPlay:WINDow<No>:STATe ON
DISPlay:WINDow<No>:TRACe:FEED "<Trc_name>"
```

## Delete Channel

Deletes the current channel including all traces assigned to the channel and removes all display elements related to the channel from the diagram area. *Delete Channel* 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.

To restore a channel that was unintentionally deleted, use the *Undo* function.

**Remote control:** `CONFigure:CHANnel<Ch>[:STATe] OFF`

## Channel Manager

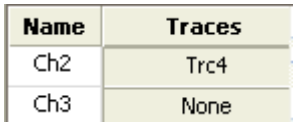
Opens a dialog to perform the actions in the *Channel Select* menu systematically and rename channels.

All existing channels of the current setup are listed in a table; see below. Below the table the *Trace Manager* provides the following buttons:

- *Add Channel* 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 Channel* deletes the channel selected in the drop-down menu. 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.

**Columns in the *Channel Manager* table**

The channel table contains several editable (white) or non-editable (gray) columns.



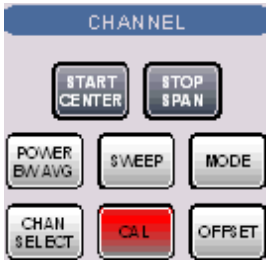
- *Channel* indicates the current channel name. The default names for new channels are Ch<n> where <n> is a current number.
- *Traces* indicates the names of all traces assigned to the channel.

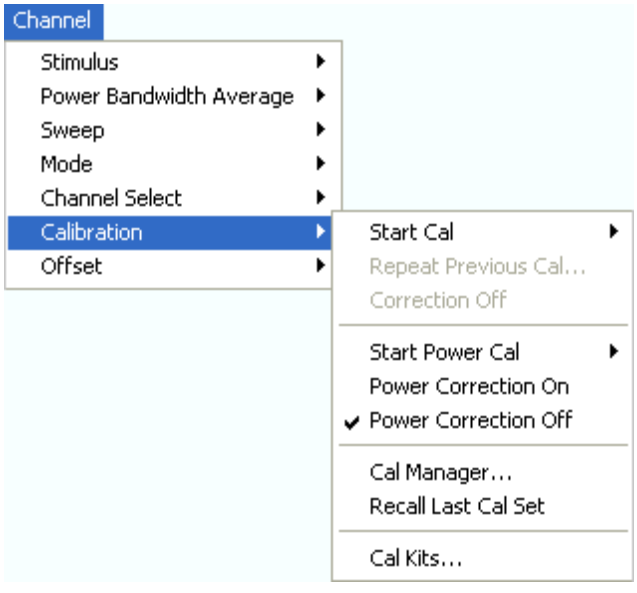
**Remote control:**
```
CONFigure:CHANnel<Ch>:CATalog?
CONFigure:CHANnel<Ch>:NAME
CONFigure:CHANnel<Ch>:NAME:ID?
CONFigure:CHANnel<Ch>[:STATe]
```

## Calibration

The *Calibration* menu provides all functions that are necessary to perform a system error correction (calibration) or a power calibration.

For an introduction to calibration and calibration types refer to section Calibration Overview in the *System Overview* chapter. See also the *Data Flow* overview in the *System Overview* chapter.

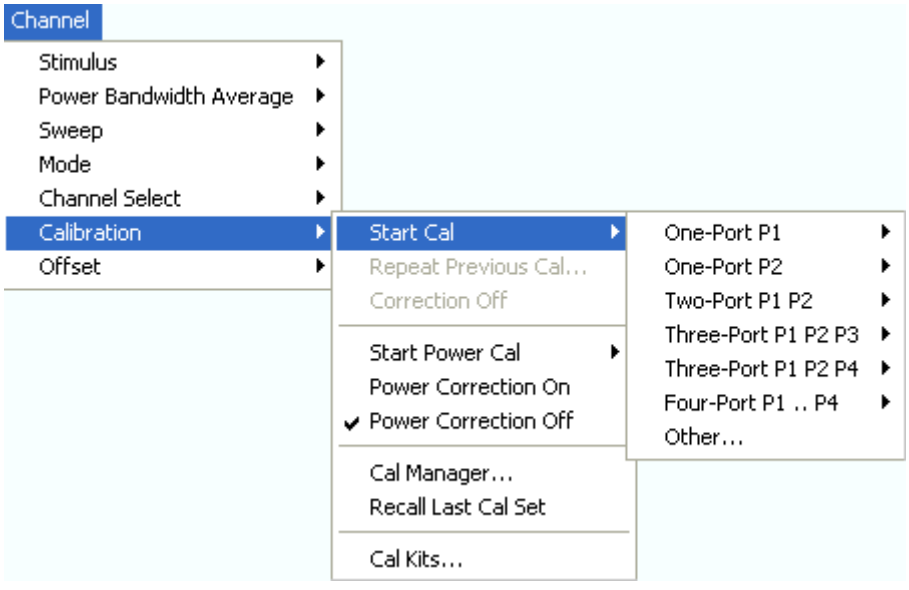



- *Start Cal* opens a submenu to select a new calibration and start the calibration wizard.
- *Repeat Previous Cal* reopens the wizard to repeat and optimize the previous calibration.
- *Correction Off* activates or deactivates the system error correction in the active channel.
- *Power Cal* opens a submenu controlling the source and receiver power calibration.
- *Power Correction On* (re-)applies the current power calibration in the active channel.
- *Power Correction Off* disables the power calibration in the active channel.
- *Cal Manager* opens a dialog to store system error correction data to a Cal Pool and to assign correction data to channels.
- *Recall Last Cal Set* loads and activates the setup for which the last calibration was performed.
- *Cal Kits* opens a dialog to manage the calibration kits in use, add new kits and import or export kits.

## Start Cal

The *Start Cal* submenu selects the calibrated ports and the calibration type.




The calibration types depend on the number of test ports of the analyzer. For a four-port unit:

- *One-Port P1* and *One Port P2* open submenus to select a one-port calibration at test ports PORT 1 and PORT 2, respectively.
- *Two-Port P1 P2* opens a submenu to select a two-port calibration at test ports PORT 1 and PORT 2.
- *Three-Port P1 P2 P3* opens a submenu to initiate a three-port (TOSM) calibration at test ports PORT 1, PORT 2 and PORT 3.
- *Four-Port P1 .. P4* opens a submenu to initiate a four-port (TOSM) calibration at test ports PORT 1, PORT 2, PORT 3 and PORT 4.
- *Other...* opens the calibration wizard to select a calibration type for an arbitrary number of ports.

Use *Other...* if you wish to perform several calibrations in a single measurement sequence and with a minimum of effort. The *Other...* calibration wizard allows you to measure a particular standard and reuse the results for different calibration types.

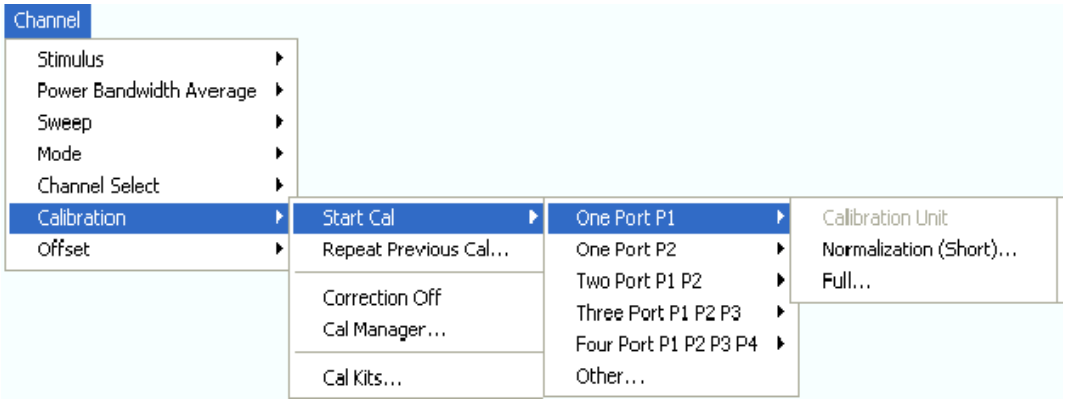
One-port to 4-port calibrations can be performed automatically using the Calibration Unit (accessory R&S ZV-Z51/-Z52). For calibrations with more than 4 ports use the calibration unit R&S ZV-Z58.

**Remote control:**

```
[SENSe<Ch>:]CORRection:COLLect:METHod:DEFine
'<cal_name>', REFL | RSHort | FOPort | FRTRans | OPTPort | TOSM |
TOM | TRM | TRL | TNA | UOSM,
<port_no>[,<port_no>][,<port_no>][,<port_no>]
```

### One Port P1, One Port P2

Opens a submenu to select a one-port calibration at test ports PORT 1 or PORT 2. The two submenus are identical:




One-port calibrations can be performed automatically or manually.

- *Calibration Unit* starts an automatic full one-port calibration. This function is enabled while the Calibration Unit is connected.

The following menu commands calls up the start dialog of the calibration wizard to start a manual calibration:

- *Normalization (Short)* initiates a normalization using a short standard.
- *Full* initiates a full one-port calibration.

**Remote control:**

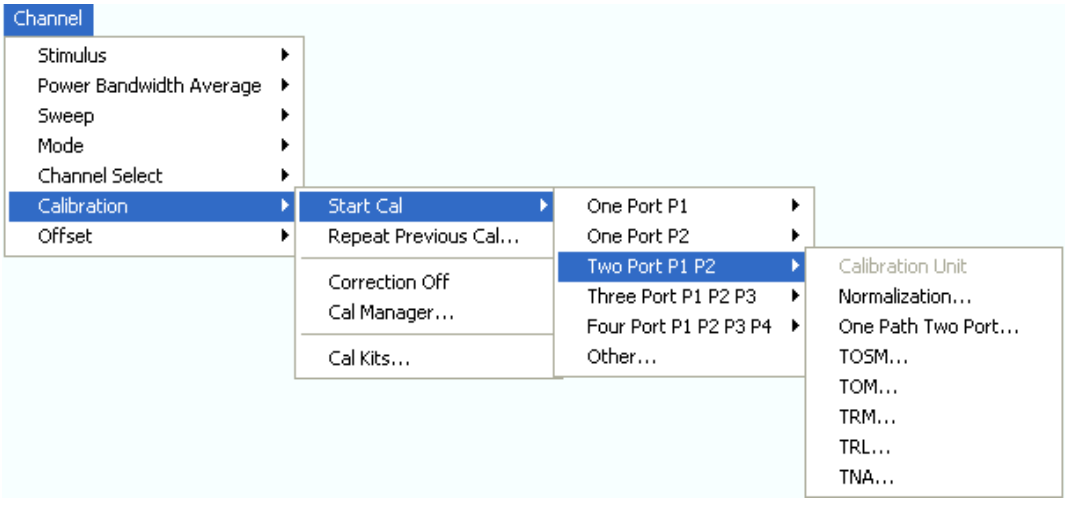
`[SENSe<Ch>:]CORRection:COLLect:AUTO ' ', <port_no>` (for matching analyzer and cal unit ports)

`[SENSe<Ch>:]CORRection:COLLect:AUTO:PORTs` (for arbitrary pairs of ports)

```
[SENSe<Ch>:]CORRection:COLLect:METHod:DEFine
"<cal_name>", REFLshort | FOPort , <port_no>
```

## Two Port P1 P2

Opens a submenu to select a two-port calibration at test ports PORT 1 and PORT 2:




Two-port calibrations can be performed automatically or manually.

- *Calibration Unit* starts an automatic tow-port TOSM calibration. This function is enabled while the Calibration Unit is connected.

The following menu commands calls up the start dialog of the calibration wizard to start a manual calibration:

- *Normalization...* initiates a normalization using a through standard.
- *One Path Two Port* initiates a full one-port calibration.
- *TOSM* initiates a Through-Open-Short-Match (12-term) calibration
- *TOM* initiates a Through-Open-Match calibration
- *TRL* initiates a Through-Reflect-Line calibration
- *TNA* initiates a Through-Network-Attenuation calibration

Use *Start Cal – Other...* to select a *UTOM* calibration.

**Remote control:**

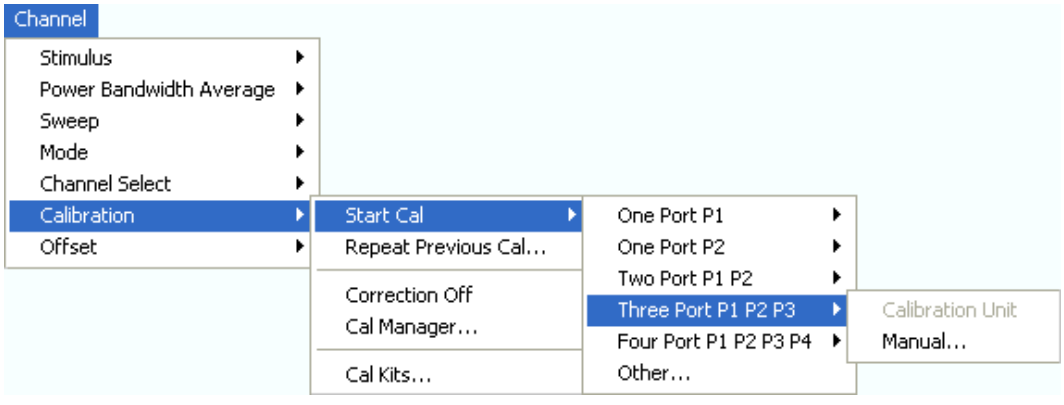
`[SENSe<Ch>:]CORRection:COLLect:AUTO ' ', <port_no>` (for matching analyzer and cal unit ports)

`[SENSe<Ch>:]CORRection:COLLect:AUTO:PORTs` (for arbitrary pairs of ports)

```
[SENSe<Ch>:]CORRection:COLLect:METHod:DEFine
"<cal_name>", FRTRans | OPTPort | TOSM | TOM | TRL | TNA,
<port_no>,<port_no>
```

## Three Port P1 P2 P3

Opens a submenu to select a three-port TOSM calibration at test ports PORT 1, PORT 2 and PORT 3:




Three-port calibrations can be performed automatically or manually.

- *Calibration Unit* starts an automatic 3-port TOSM calibration. This function is enabled while the Calibration Unit is connected.
- *Manual...* calls up the start dialog of the calibration wizard to initiate a 3-port TOSM calibration.

Use *Start Cal – Other...* to select other 3-port calibration types.

**Remote control:**

`[SENSe<Ch>:]CORRection:COLLect:AUTO ' ', <port_no>` (for matching analyzer and cal unit ports)

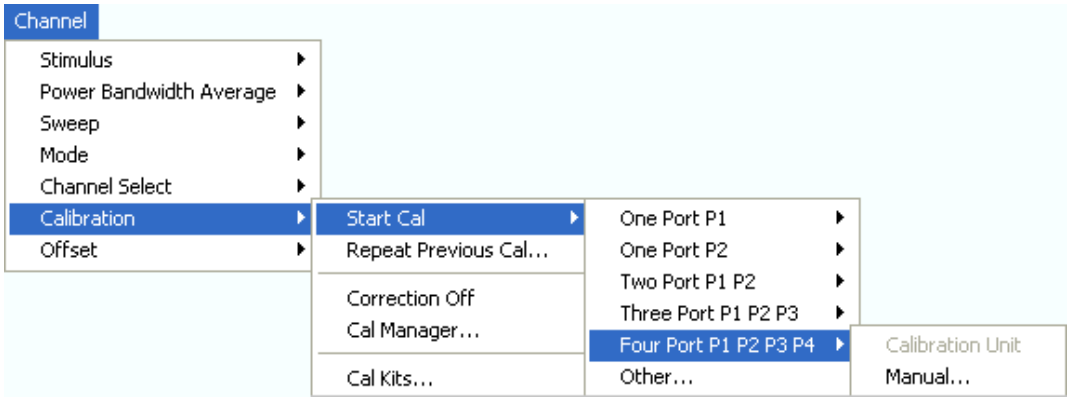
`[SENSe<Ch>:]CORRection:COLLect:AUTO:PORTs` (for arbitrary pairs of ports)

`[SENSe<Ch>:]CORRection:COLLect:METHod:DEFine`

```
"<cal_name>", TOSM, <port_no>, <port_no>, <port_no>
```

### Four Port P1 P2 P3 P4

Opens a submenu to select a four-port TOSM calibration at test ports PORT 1 to PORT 4:




Four-port calibrations can be performed automatically or manually.

- *Calibration Unit* starts an automatic 4-port TOSM calibration. This function is enabled while the Calibration Unit is connected.
- *Manual...* calls up the start dialog of the calibration wizard to initiate a 4-port TOSM calibration.

Use *Start Cal – Other...* to select other 4-port calibration types.

**Remote control:**

```
[SENSe<Ch>:]CORRection:COLLect:AUTO ' ', <port_no>
```
(for matching analyzer and cal unit ports)

```
[SENSe<Ch>:]CORRection:COLLect:AUTO:PORTs
```
(for arbitrary pairs of ports)

```
[SENSe<Ch>:]CORRection:COLLect:METHod:DEFine
"<cal_name>", TOSM, <port_no>, <port_no>, <port_no>, <port_no>
```

## Guided Calibration

The analyzer provides a calibration wizard for each calibration type. The guided calibration consists of the following steps:

19. *Select Physical Port Connectors* and calibration kits at all calibrated ports.
20. *Compile Calibrations:* Select a calibration type and the physical ports to be calibrated (skipped for predefined calibrations).
21. *Measure Standards:* Acquire measurement data for all standards required for the selected calibration type.
22. Calculate the system error correction data (error terms) from the measurement data of the standards and apply the result to the active channel.

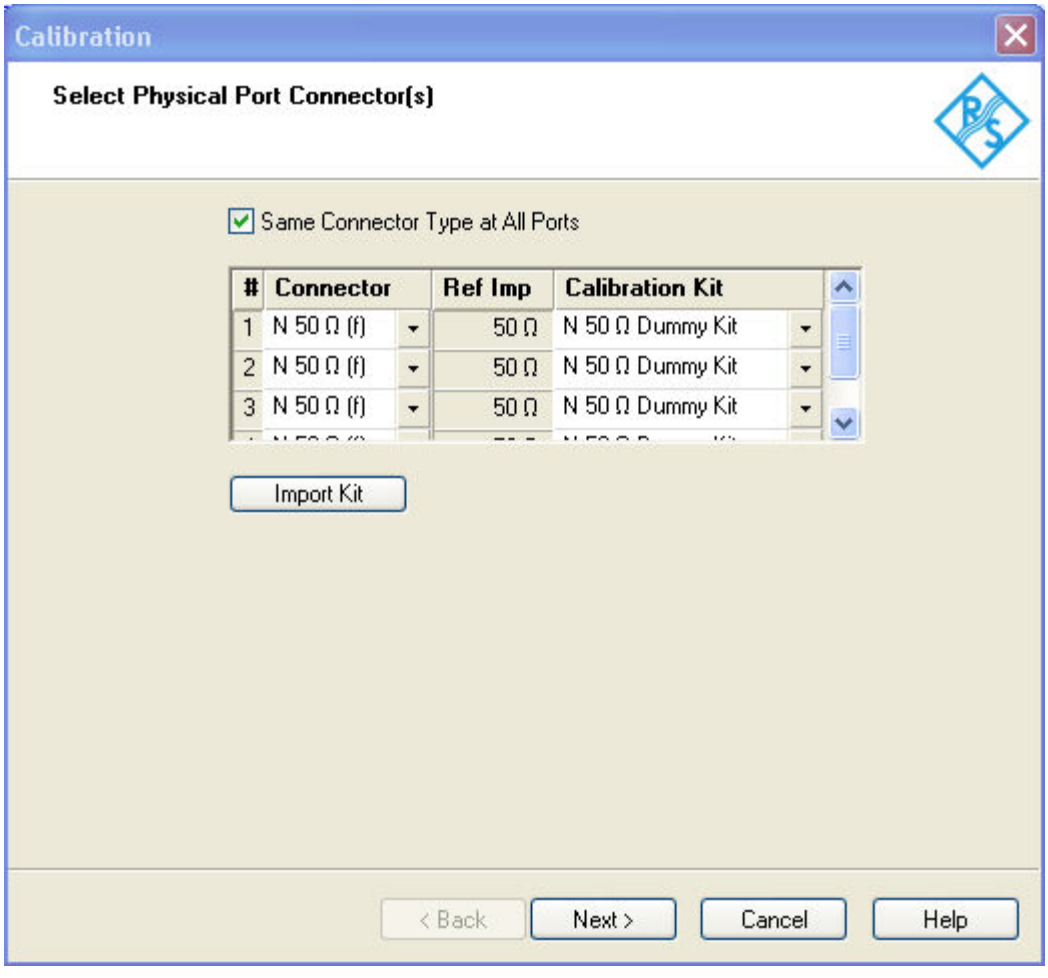
A successful calibration will supersede the previous calibration, discarding all previous system

error correction data. To keep older correction data you can transfer them into a *Cal Pool* using the *Calibration Manager.*

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.

## Select Connectors

The first dialog of the calibration wizard displays a table to select the connectors and calibration kits for all calibrated physical ports.




The table contains the following rows:

- ***Physical Port Number #***

  The ports (and therefore the number of table rows) are determined by the active calibration type selected in the *Start Cal* submenu. For a user-defined calibration *(Start Cal – Other...)* all measurement ports of the analyzer are displayed regardless of their port configuration.

  NOTE To perform a system error correction, the physical port must be available as a source and receive port. This condition is met if it is selected as a measured port in the *Balanced and Measured Ports* dialog *(Meas* is on).

- ***Connector***

  provides a drop-down list to select the connector type An (f) behind the connector type denotes female connectors, an (m) denotes male connectors. Symmetric (sexless) connectors (e.g. PC7) are not labeled. User-defined connectors can be added or removed in the *Available Connector Types* dialog, which is opened from the *Channel – Calibration – Calibration Kits* dialog. at the port and its gender. If *Same Connector at All Ports in Table* is active, the connector types at all ports (but not their gender) are always adjusted to the current selection.

- *Ref. Imp.* shows the reference impedance for the selected connector.

- ***Calibration Kits***

  provides a drop-down list to select a calibration kit. The list contains all calibration kits available 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 analyzer automatically assigns the new kit to all ports with the same connector type.

  ***Completness:*** *The selected calibration kit must contain all standards needed for the active calibration type. If it doesn't the analyzer displays an error message.*

  ***Waveguide cutoff:****If a user-defined waveguide connector is assigned to one of the calibrated ports, then the start frequency of the active channel must be above the waveguide cutoff. Otherwise the analyzer displays an error message.*

*Import Kit* opens the *Import Calibration Kit* dialog to load and (if desired) activate a cal kit file. *Next >* opens the second dialog of the wizard to continue the calibration procedure:

- In a user-defined *(Other...)* calibration, the *Compile Calibrations* dialog is called up.
- In a predefined calibration, *Compile Calibrations* is skipped and the *Measure Standards* dialog is called up.

*If the calibrated channel has already been assigned to a cal group, the correction data overwrites the cal group data, so the new calibration will affect all channels assigned to the cal group. The network analyzer generates a notice message "New calibration will overwrite cal pool!" when opening the Select Physical Port Connectors dialog.*

**Checks on switching to the next dialog**

When the *Next>* buttonis pressed the analyzer checks the calibration kits and the matching of the calibration standards and possibly displays a notice box (confirm with *OK).* This happens:

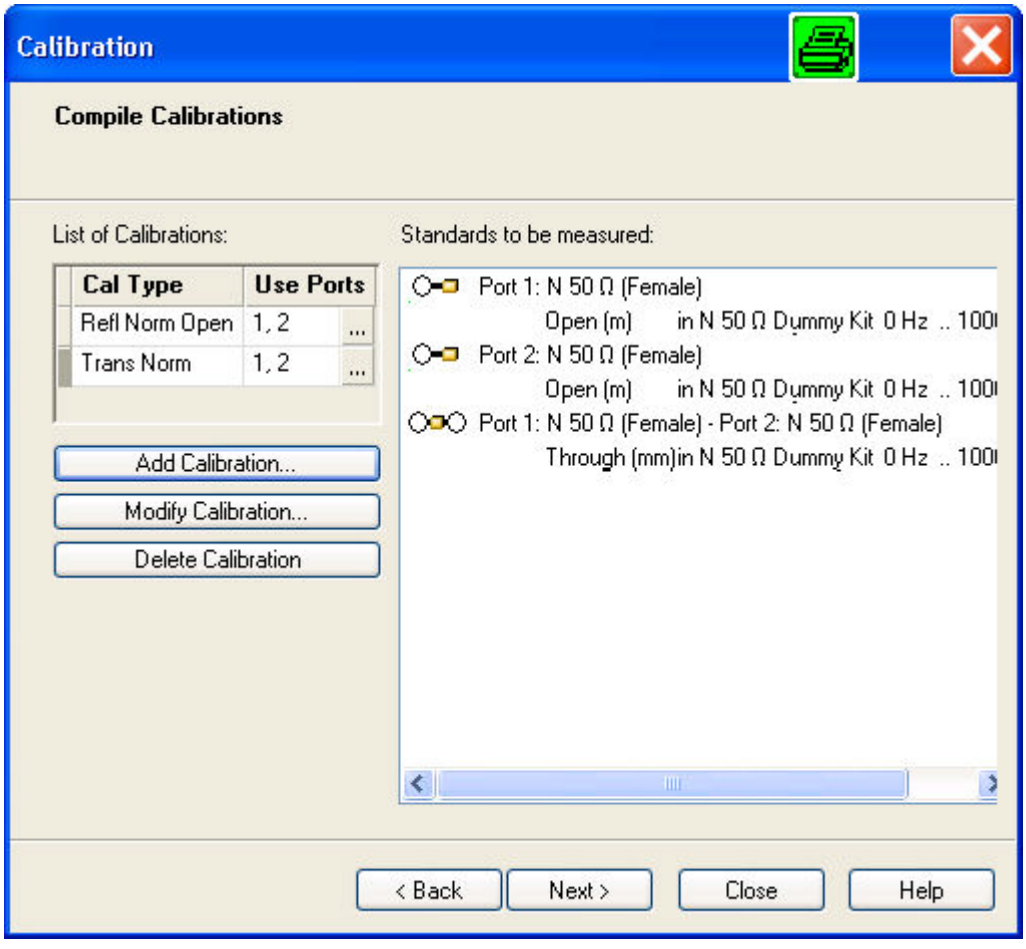
- If one of the calibration kits is described by ideal kit parameters or typical values.
- If a predefined 7-term calibration method TXX *(TOM, TRL, TNA, TRM)* is selected but one of the calibration standards is mismatched. A standard is assumed to be mismatched if it is described by S-parameter data (*.snp file) with a reflection factor > –30 dB.

**Remote control:**
```
[SENSe<Ch>:]CORRection:COLLect:CONNection<port_no>
[SENSe<Ch>:]CORRection:COLLect:SCONnection<port_no>
[SENSe<Ch>:]CORRection:CKIT:<conn_type>:SELect "<Ckit_Name>"
```

## Compile Calibrations

The second dialog of the calibration wizard appears for user-defined *(Other...)* calibrations only. It is used to compile a list of calibration types and physical ports in order to perform all calibrations in a single measurement sequence. A measurement of a particular standard that is required for different

calibration types can be reused.




The selected calibrations and ports appear in the *List of Calibrations.* A (N) behind the ports number denotes a node port (see *Add Calibration* dialog). The list of *Standards to be Measured* is automatically generated according to the *List of Calibrations.*

The dialog provides three buttons to extend or modify the *List of Calibrations:*

- *Add Calibration...* opens the *Add Calibration* dialog to add a new calibration to the list.
- *Modify Calibration...* opens the *Add Calibration* dialog to modify the selected calibration.
- *Delete Calibration* removes the selected calibration from the list.

Two buttons at the bottom of the dialog move back and forward in the calibration wizard:

- *< Back* opens the *Select Physical Port Connectors* dialog.
- *> Next* opens the *Measure Standards* dialog.

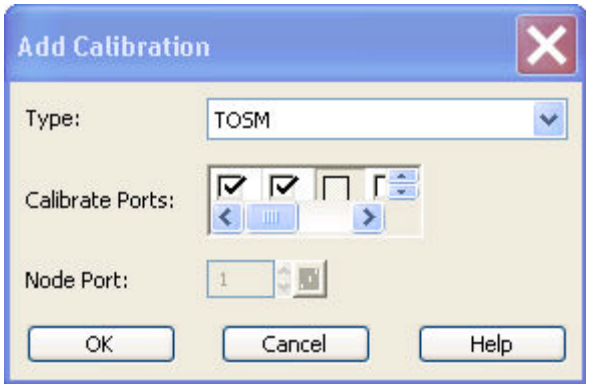
If one of the calibration kits is described by ideal kit parameters or typical values, the analyzer displays a notice box.

**Remote control:**

```
[SENSe<Ch>:]CORRection:COLLect:METHod:DEFine
"<cal_name>", REFLshort | FOPort | FRTRans | OPTPort | TOSM | TOM
| TRM | TRL | TNA | UOSM,
<port_no>[,<port_no>][,<port_no>][,<port_no>]
```

## Add Calibration

The *Add Calibration* dialog adds a new calibration to the list in the *Compile Calibrations* dialog or modifies an existing calibration.




- *Type* contains a drop-down list to select a calibration type.
- *Calibrate Ports* provides a list of all test ports of the analyzer (no external device ports). Ports with unsuitable port configuration (see *Select Connector – Physical Port Number #)* are grayed.
- *Node Port* is available for *One Path Two Port* calibrations where it selects the fully calibrated port. All ports selected in the *Calibrate Ports* table can be used as node ports.

**Checks on closing the dialog**

When the *OK* buttonis pressed the analyzer checks the calibration kits and the selected calibration (s) and possibly displays a notice box (confirm with *OK)* or astatus message. This happens:

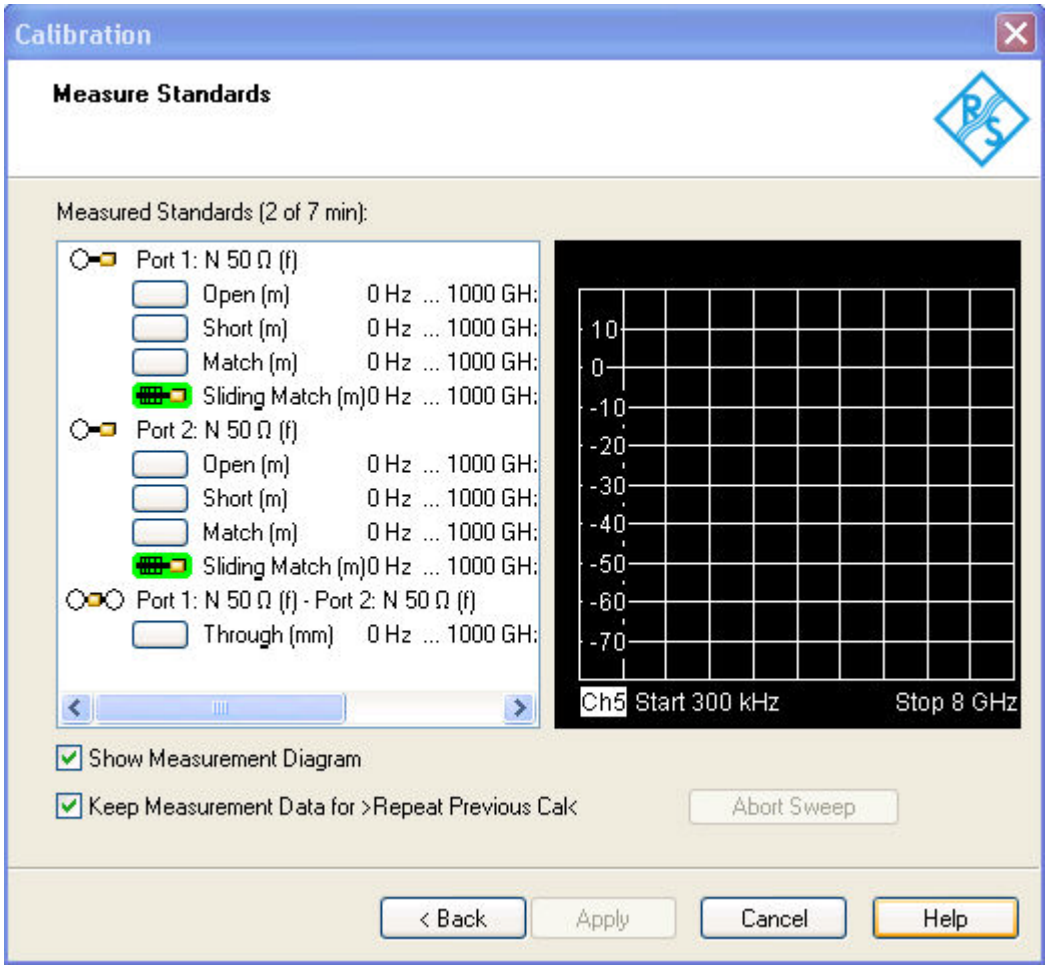
- If one of the calibration kits is described by ideal kit parameters or typical values.
- If one of the selected calibration types is redundant. A calibration is redundant if another selected calibration provides more complete correction data. Redundant calibrations are deleted from the list of calibrations.

**Remote control:**

```
[SENSe<Ch>:]CORRection:COLLect:METHod:DEFine
"<cal_name>", REFLshort | FOPort | FRTRans | OPTPort | TOSM | TOM
| TRM | TRL | TNA | UOSM,
<port_no>[,<port_no>][,<port_no>][,<port_no>]
```

## Measure Standards

The last dialog of the calibration wizard is used to perform the necessary measurements of standards and to calculate the correction data.




*Measured Standards* displays the list of measured standards compiled in the previous dialogs. The list is complete: All standards must be measured to perform the selected calibrations.

**Structure of the *Measured Standards* list**

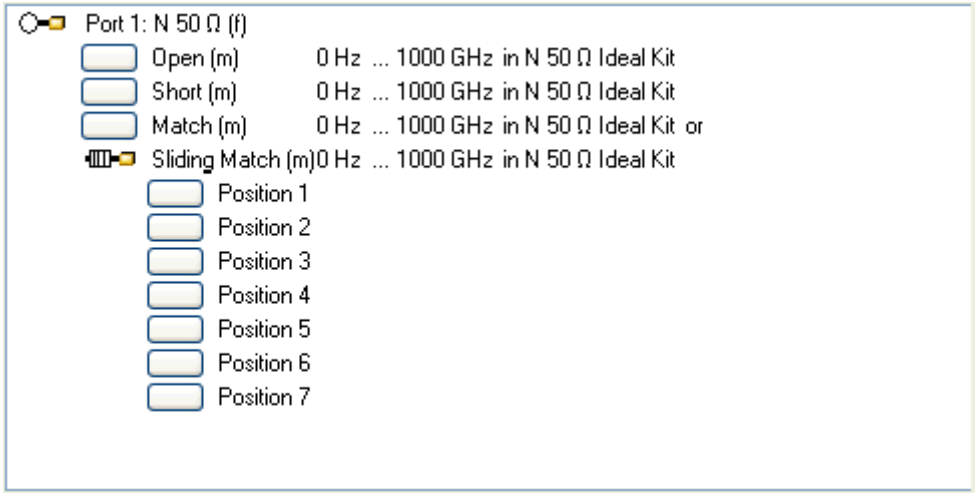
The list of measured standards has a tree structure.

- The first level contains all physical ports where one-port (reflection) measurements are required and all physical port combinations where two-port (transmission) measurements must be performed.
- The second level contains check boxes for the standards to be measured at each port or port combination.
- For a sliding match, a third level contains check boxes for the different positions of the sliding element.

Double-click a physical port symbol to expand or collapse the list.

**Calibrations using a match or sliding match**

If the calibration kit contains a sliding match standard, the *Sliding Match* appears in the *Measured Standards* list whenever the selected calibration type requires a *Match.* A click on the node expands the check boxes for the different positions of the load element. The number of different positions is defined in the *User Interface* tab of the *System Configuration* dialog.




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 load 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 – 6 can improve the accuracy. Moreover it is recommended to use the predefined load positions of the standard.

The calibration is valid *(Apply* is available) 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 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 will provide better results than the match within its specified frequency range.

TRL extensions: calibration with two lines, combination with TRM

Checking one of the boxes in the list causes the analyzer to stop the measurement in all channels except the active one and measure the standard according to the active channel settings. The progress of the calibration sweep and the result can be monitored in the diagram. In case of an error (e.g. if the measurement result shows that the calibration standard was not connected properly), *Abort Sweep* immediately terminates the sweep.

After completing the sweep the analyzer generates a short sound and a green checkmark appears in the checkbox. Measurements can be repeated as often as desired. Newer results overwrite older measurement data.

*Most channel settings including the trigger settings remain valid for calibration sweeps and power calibration sweeps. To start the calibration sweeps without delay select Free Run trigger mode.*

**Checks for the calibration sweep**

If the generator frequency range of the active ports exceeds the validity range of the standard model (defined by *Min. Freq.* and *Max. Freq.* in the *Add/Modify Standard* dialog) the analyzer displays a notice

box (confirm with *OK).*

The *Measure Standards* dialog provides further controls:

- *Show Measurement Diagram* displays or hides the diagram to the right of the list of *Measured Standards*. Hiding the diagram leaves more space for displaying the characteristics of the measured standards.
- *Keep Measurement Data for >Repeat Previous Cal<* causes the raw measurement data of the standards to be stored after the calibration is completed. This enables the *Repeat Previous Cal...* command, which can be used to optimize a previous calibration without repeating the measurement of all standards. If *Keep Measurement Data...* is not active, then the raw measurement data of the standards is deleted and the analyzer only stores the system error correction data. Deleting the raw data saves disk space.
- *Apply* is enabled as soon as data has been acquired for all standards. The button starts the calculation of the system error correction data and closes the calibration wizard. The current instrument settings are stored with the correction data. To avoid incompatibilities, older system error correction data is deleted unless it has been transferred into a *Cal Pool* using the *Calibration Manager.* If an unknown through standard is measured, *Apply* opens an additional dialog to check the calculated length offset parameters; see *Unknown Through Standard*.

The *Keep Measurement Data for >Repeat Previous Cal<* setting is valid for the current calibration only. To activate this function in general, use the parameter in the *User Interface* tab of the *Sytem Config.* dialog (menu *System – System Config*).

**Checks during the calculation of correction data**

Incompatibilities between the selected calibration type, the standards and the channel settings may cause the calibration to be inaccurate. The analyzer auto-detects potential sources of errors and displays appropriate, self-explanatory notice boxes.

**Remote control:**
```
[SENSe<Ch>:]CORRection:COLLect[:ACQuire]:RSAVe
[SENSe<Ch>:]CORRection:COLLect[:ACQUire]:SELected
[SENSe<Ch>:]CORRection:COLLect:SAVE
[SENSe<Ch>:]CORRection:COLLect:DELete ["<cal_name>"]
```

## TRL Extensions

The frequency range for a TRL calibration is restricted due to singularities in the system of equations to be solved. Singularities occur whenever the length difference $\Delta L$ between the through and the line is an integer multiple of half of the wave length:

$$\Delta L \neq n \frac{\lambda}{2}$$

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 TRL calibration with two different 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

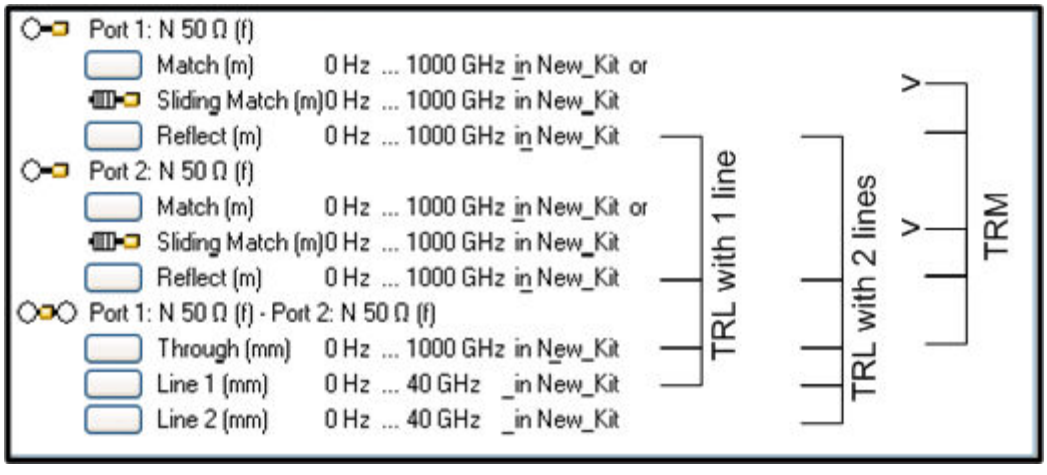
approx. 1:64.

- In the low-frequency domain where TRL becomes inaccurate, replaced TRL by TRM calibration.

The methods can be combined or used separately. The *Measured Standards* list for TRL calibration is extended if the calibration kit in use contains the necessary standards:

- A 2-line calibration requires two different lines of matching gender. The lines must be measured between any combination of two ports.
- A TRM extension at low frequencies requires either a match or a sliding match standard. The standard must be measured at each port.

The complete *Measured Standards* list for a two-port calibration is shown below. The calibration is valid as soon as the standards for *TRL with 1 line* have been measured. The extensions are applied automatically if the necessary standards have been measured.




**Calibration with two lines**

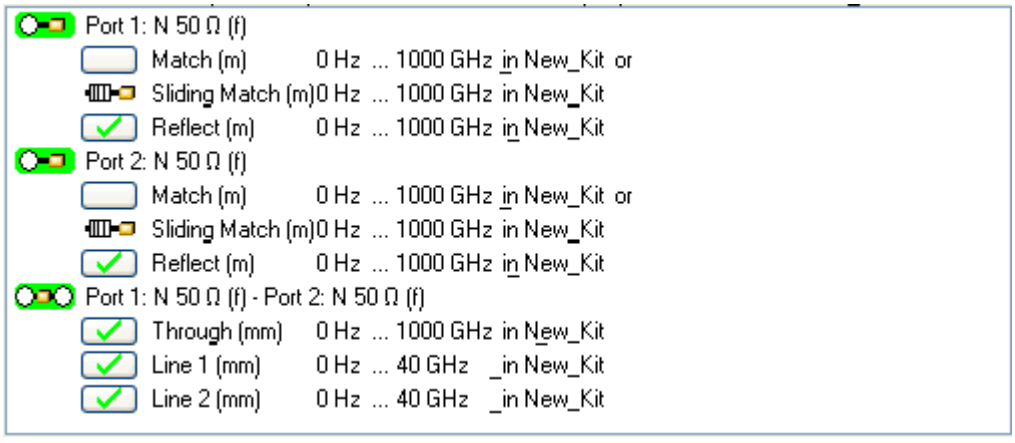
If both lines with different lengths are measured, the analyzer automatically divides the calibrated range into two segments; see below. The calibration data of the longer line is applied to the lower segment, the calibration data of the shorter line to the upper segment.

**Calculation of segments**

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 $l_{thr}$):

23. 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 $f_{long} = 4*c_0/[9*(l_{long} - l_{thr})]$.
24. 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 $f_{short} = c_0/[18*(l_{short} - l_{thr})]$.
25. The border between the two frequency segments $f_{div}$ is calculated as the geometric mean of $f_{long}$ and $f_{short}$, i.e. $f_{div} = \mathrm{sqrt}(f_{long} * f_{short})$.

The formulas are also applied if $f_{long} < f_{short}$.



A second line in the list does not mean that you have to measure both standards. If the calibrated frequency range is small enough, the calibration is valid as soon as the analyzer has acquired correction data for a single line standard. 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**

L calibration becomes inaccurate if the electrical length difference between line and through standard corresponds to a phase shift below 20°. In practice this means that TRL is only practicable above a threshold frequency $f_{TRM}$ which depends on the lengths of the line and through standards. The threshold frequency is given by:

$$f_{TRM} = c_0/[18*(l_{long} - l_{thr})]$$

where $l_{long}$ denotes the electrical length of the longer of the used line standards, $l_{thr}$ the length of the through. The analyzer assumes $l_{thr} << l_{long}$ and calculates $f_{TRM} = c_0/(18*l_{long})$. At frequencies below $f_{TRM}$, TRL calibration is automatically replaced by TRM, provided that the necessary calibration data has been acquired. For a line with $l_{long}$ = 16.666 cm, the threshold frequency is $f_{TRM}$ = 100 MHz.

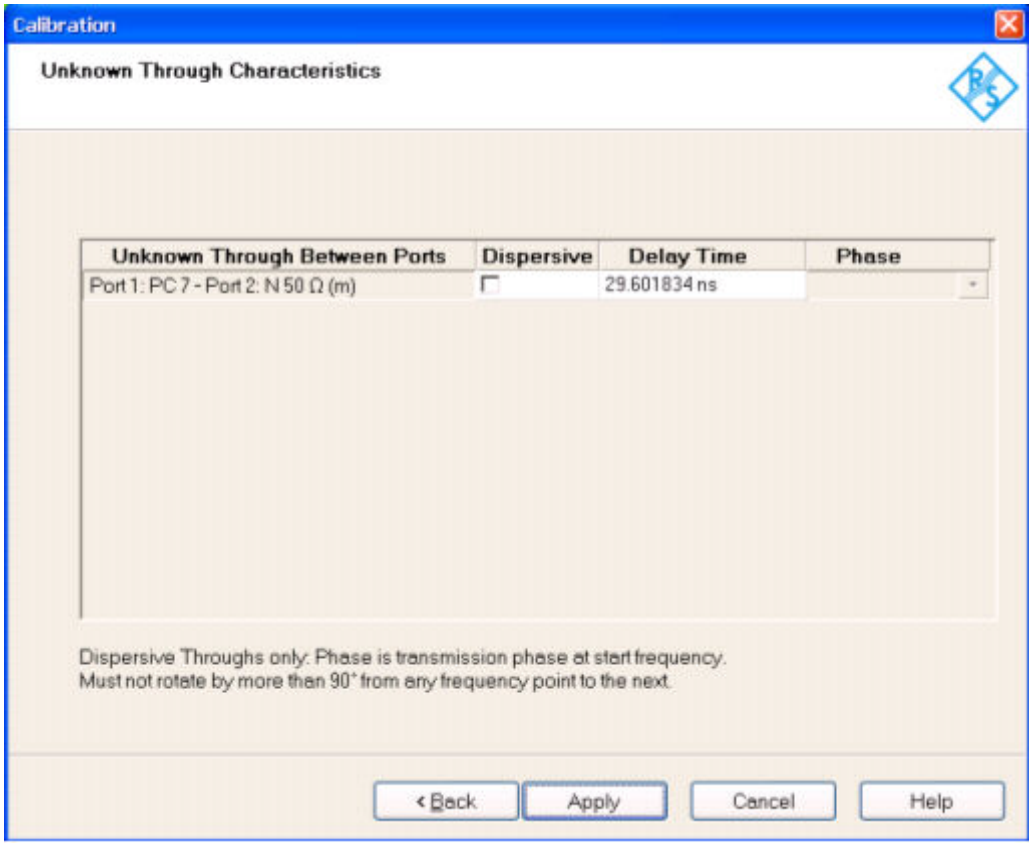
**TRL calibration with low-frequency TRM extension**

**Accuracy conditions for the line(s)**

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.

## Unknown Through Standard

If an unknown through standard (or an unknown reciprocal mixer; see Mixer Vector Cal) is measured, the *Apply* button in the *Measure Standards* dialog opens an additional dialog.




The *Unknown Through Characteristics* dialog shows the delay time or transmission phase that the analyzer determined during the calibration sweep.

- For a non-dispersive standard *(Dispersive* check box cleared), the *Delay Time* can be determined unambiguously, provided that the transmission phase difference between two consecutive sweep points is below 90 deg (make sure that the spacing between the sweep points is small enough). In this case it is sufficient to press *Apply* in order to calculate the system error correction data and close the calibration wizard.
- For a dispersive standard *(Dispersive* check box selected), the delay time is frequency-dependent, however, the analyzer can determine two solutions for the transmission phase at the start frequency of the calibration sweep. The two solutions differ by 180 deg; the right solution must be selected from a drop-down list.

If you are not sure which one of the two phases for a dispersive standard is correct, select one, press *Apply* and check the measured result. If the transmission phase looks incorrect, use *Channel – Calibration – Repeat Previous Cal...* to select the alternative solution.

*The delay time or phase determined by the analyzer can be corrected manually e.g. by entering a value derived from the mechanical length of the standard. Doubtful delay times or phases are displayed with a question mark.*

**Remote control:** `[SENSe<Ch>:]CORRection:COLLect[:ACQuire]`
`[SENSe<Ch>:]CORRection:COLLect[:ACQUire]:SELected`

## Repeat Previous Cal

Reopens the *Measure Standards* dialog of the calibration wizard to repeat and optimize the previous calibration without necessarily repeating the measurement of all standards.

*Repeat Previous Cal* is active only if *Keep Measurement Data for >Repeat Cal<* in the *Measure Standards* dialog was enabled when the last calibration for the active channel was successfully completed.

**Additional conditions for repeating a calibration**

Changing the channel settings can cause incompatibilities between the current instrument state and the previous calibration. In particular the following conditions must be met:

- The configuration of all ports needed must be correct (see *Select Connector– Physical Port Number #)*.
- The *Sweep Type* must be unchanged.

If incompatible settings occur the analyzer displays an error message and does not repeat the calibration.

## Recall Last Cal Set

Loads and activates the setup 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:\Rohde&Schwarz\Nwa\Calibration\RecallSets* directory. A message box pops up if the directory is empty, e.g. because no calibration was performed yet.

## Correction Off

Activates or deactivates the system error correction in the active channel. *Correction Off* is active only if a valid system error correction is assigned to the active channel; see *Cal State* in the *Calibration Manager.*

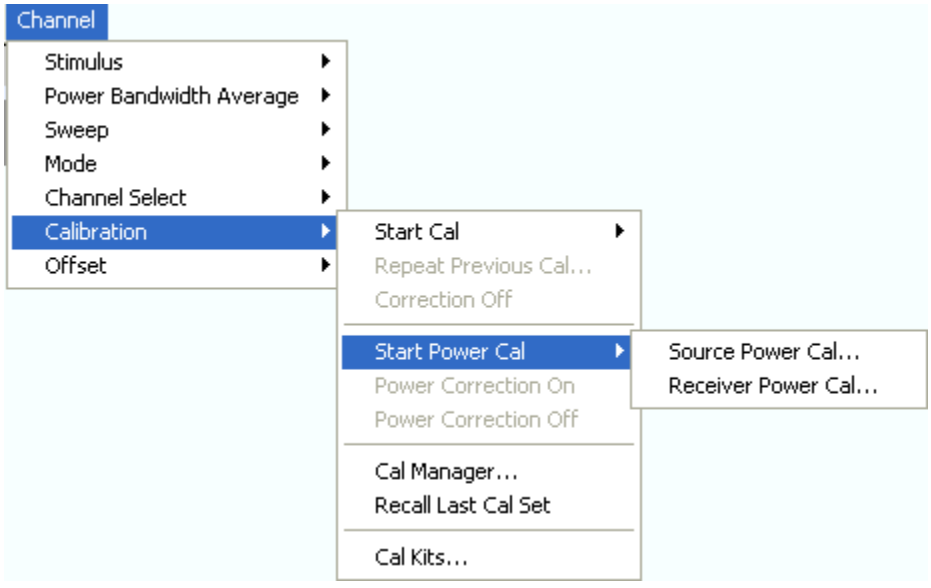
*A red label Cal Off ! appears behind the trace list if the system error correction is switched off; see also Calibration Overview.*

**Remote control:** `[:SENSe<Ch>]:CORRection:STATe ON | OFF`

## Power Calibration

The *Power Calibration* submenu configures and starts a source or receiver power calibration.

*In true differential mode only the source power calibration results are used, no receiver power calibration is possible.*




- *Source Power Cal...* opens the configuration dialog for a source power calibration.
- *Receiver Power Cal...* opens the configuration dialog for a receiver power calibration.

*Power correction data can be stored to a cal group and applied to several channels; see description of the Calibration Manager.*
*In remote control it is possible to read, modify and reload power calibration data. Refer to the description of the commands* `SOURce<Ch>:POWer<Pt>:CORRection:DATA` *and* `[SENSe<Ch>:]CORRection:POWer<port_no>:DATA`.

**Remote control:** `SOURce<Ch>:POWer:CORRection...`
`[SENSe<Ch>:]CORRection:POWer...`

## Source Power Cal...

Opens the configuration dialog for a source power calibration.

**Purpose of a source power calibration**

A source power calibration ensures accurate power levels of the generated wave at an arbitrary reference plane in the measurement path. The calibrated reference plane is typically the input of the DUT.

In a frequency sweep, the power at the reference 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 reference 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 reference 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.

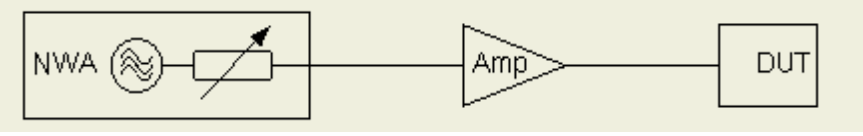
The correction data acquired in a frequency or power sweep is re-used if a *Time* or *CW Mode* sweep is activated.

**Calibration procedure**

The source power 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 reference plane or to any other point in the test setup where the signal power is known to be proportional to the power at the reference plane.

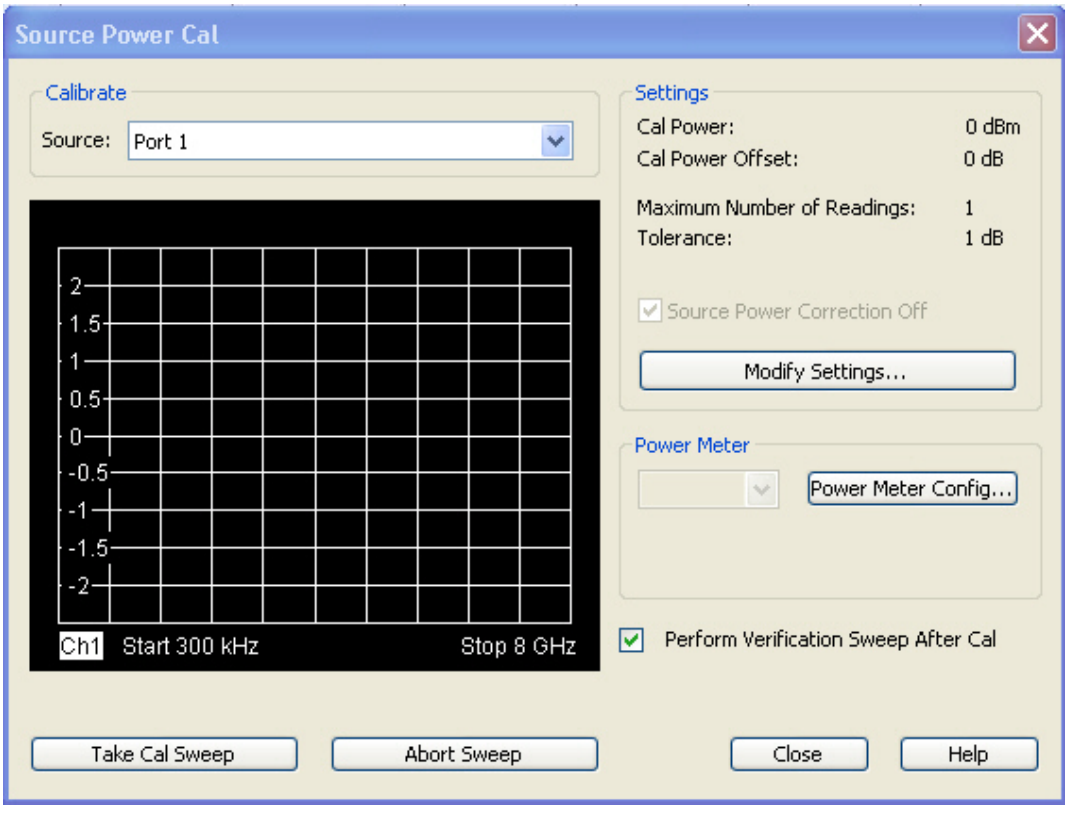
The source power calibration involves several steps:

26. The analyzer performs a first calibration sweep at the source power that is likely to produce the target power (*Cal Power)* at the reference plane. A known attenuation or gain at the source port and in the signal path between the source port and the reference plane is taken into account.




27. The power measured at the reference plane is displayed in the calibration sweep diagram and compared to the *Cal Power.* The comparison serves as an input for the correction of the source power.
28. Steps 1 and 2 are repeated for a specified *Number of Readings* or after the deviation between the measured power at the reference plane and the *Cal Power* is below a specified *Tolerance.*
29. After the last calibration sweep, it is possible to perform an additional *Verification Sweep* in order to check the accuracy.

After a power calibration one can expect the power at the reference plane to be within the range of uncertainty of the power meter. After a change of the sweep points or sweep range, the analyzer interpolates or extrapolates the calibration data.



The *Source Power Cal* dialog provides the following control elements:

- *Calibrate – Source* contains all analyzer ports or external generators which can be used as a source for the calibrated wave quantity. The list contains all analyzer ports *Port 1* to *Port n*. Generators *(Gen 1, Gen 2, ...)* must be configured explicitly in the *System Configuration – External Generators* dialog before they appear in the list.
- *Take Cal Sweep* initiates a calibration sweep for the selected source. The progress of the sweep is monitored in the calibration sweep diagram. After the necessary calibrations have been completed, a message in the trace list indicates the status of the power calibration.
- *Abort Sweep* stops a running sweep. The data acquired in the aborted sweep is discarded.
- The *Settings* panel displays the essential parameters of the source power calibration, to be defined in the *Modify Source Power Cal Settings* dialog.
- *Source Power Correction Off* can be selected/deselected as soon as power calibration data has been acquired for the selected source. If the box is checked, the calibration data is kept in memory but not applied to the source. The *Power Correction Off* softkey disables the power correction for all receivers and sources.
- *Modify Settings...* opens the *Modify Source Power Cal Settings* dialog.
- *Power Meter* selects an external power meter that is used to measure the exact source power at the calibration point. Power meters *(Pmtr1, Pmtr2, ...)* must be configured explicitly in the *System Configuration – External Power Meters* dialog before they appear in the list.
- *Power Meter Config...* opens the *System Configuration – External Power Meters* dialog.
- *Perform Verification Sweep after Cal* enables or disables a verification sweep that the analyzer performs after the source power calibration. The sweep is also displayed in the calibration sweep diagram. Its purpose is to test the accuracy of the source power calibration after the final correction data has been acquired. The final power calibration results are applied starting with the first sweep after the verification sweep.

**Remote control:**

```
SOURce<Ch>:POWer<Pt>:CORRection[:ACQuire]
SOURce<Ch>:POWer<Pt>:CORRection:COLLect[:ACQuire]
SOURce<Ch>:POWer<Pt>:CORRection[:STATe]
SOURce<Ch>:POWer<Pt>:CORRection:GENerator[:STATe]
SOURce<Ch>:POWer<Pt>:CORRection:PMETer:ID
SOURce<Ch>:POWer<Pt>:CORRection:COLLect[:ACQuire]:VERification[:STATe]
SOURce<Ch>:POWer<Pt>:CORRection[:ACQuire]:VERification:RESult?
```

## Calibration Sweep Diagram

The calibration sweep diagram displays the power measured by the external power meter while a source power calibration is performed.

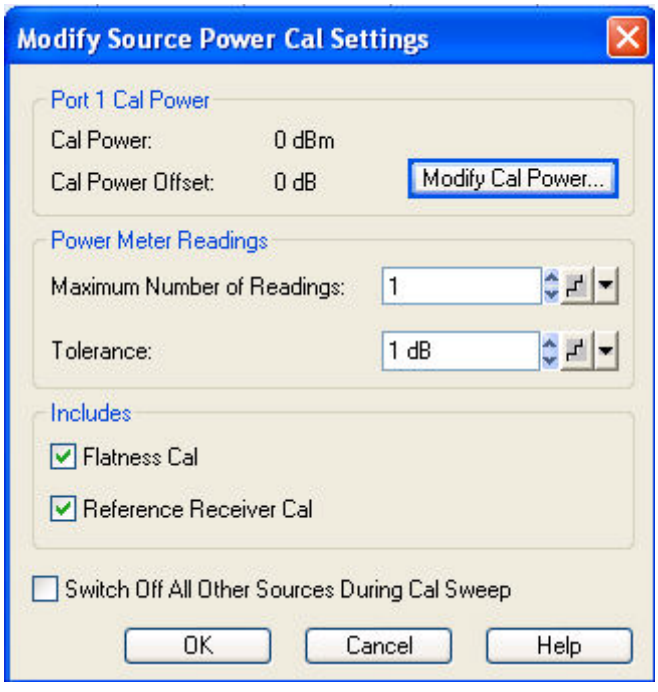
- A trace is displayed for each calibration sweep.
- If *Perform Verification Sweep after Cal* is enabled in the *Source Power Cal* dialog, an additional trace is displayed after the end of the calibration.

The diagram is automatically scaled so that the center corresponds to the selected *Cal Power.* For frequency sweeps, a horizontal trace close to the *Cal Power* indicates an accurate power calibration. For power sweeps, the ideal trace is a power ramp.

Below the sweep diagram, the status messages *Calibration in Progress/Finished* and *Verification in Progress/Finished* show the progress of the calibration procedure.

## Modify Source Power Cal Settings

The *Modify Source Power Cal Settings* dialog displays the target power for the source power calibration and defines stop criteria.




The dialog provides the following control elements:

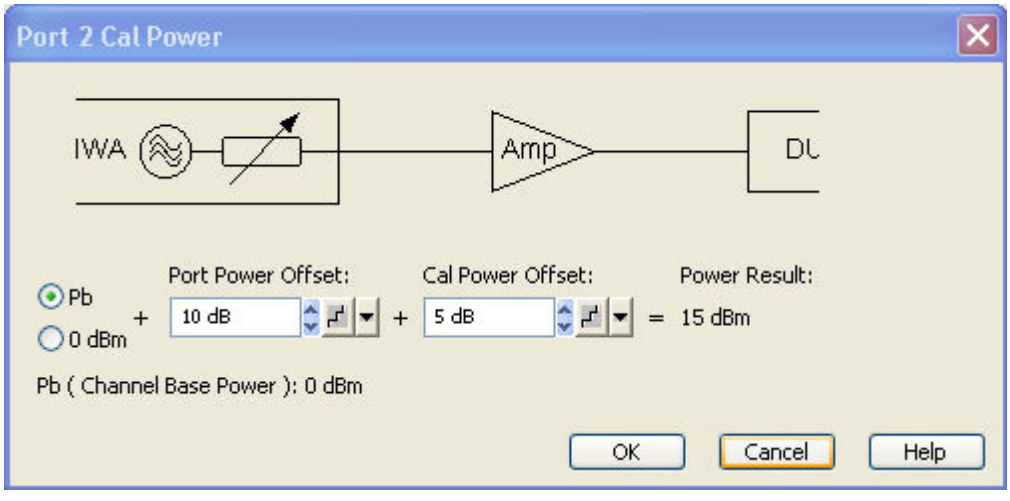
- *Port n Cal Power* (where n is the calibrated source port selected in the *Source Power Cal* dialog) shows the target power of the calibration *(Cal Power)* and the *Cal Power Offset*. Both parameters can be adjusted in the *Port n Cal Power* dialog, to be opened via *Modify Cal Power...*
- The parameters in the *Power Meter Readings* panel provide stop criteria for the calibration procedure: *Maximum Number of Readings* is a limit for the number of calibration sweeps. *Tolerance* denotes the maximum deviation of the measured power from the *Cal Power.* The calibration procedure is stopped if the *Maximum Number of Readings* is reached **or** if the measured power is within the *Tolerance*.
- *Includes – Flatness Cal* enables the iterative source power calibration with the stop criteria specified above. This corresponds to the normal source calibration procedure. If the flatness cal is disabled, the analyzer performs a single calibration sweep in order to calibrate the reference receiver (see below) but without overwriting the previous source power calibration data. The *Port n Cal Power* and *Power Meter Readings* are disabled. A frequency extension with an external converter is a typical example for a measurement where the *Flatness Cal* should be disabled.
- *Includes – Reference Receiver Cal* enables a calibration of the reference receiver of the active receiver port together with the source power calibration. The reference receiver reading is set in accordance with the power reading of the external sensor. This can replace a subsequent a-wave receiver power calibration.
- If *Switch off All Other Sources...* is checked, the power at all sources except the calibrated source is switched off during the calibration. This is advisable especially if the power calibration is intended for intermodulation measurements where different signals are combined.

*The analyzer performs either a Flatness Cal or a Reference Receiver Cal, or both. The two options must not be disabled simultaneously.*

**Remote control:**
```
SOURce<Ch>:POWer<Pt>:CORRection:NREadings
SOURce<Ch>:POWer<Pt>:CORRection:COLLect:AVERage[:COUNt]
SOURce<Ch>:POWer<Pt>:CORRection:COLLect:AVERage:NTOLerance
SOURce<Ch>:POWer<Pt>:CORRection:OSOurces[:STATe]
```

## Port n Cal Power

The *Port n Cal Power* dialog defines the target power for the source power calibration. The settings are port-specific.




The *Port n Cal Power* settings serve two different purposes:

- *Port Power Offset* defines a port-specific offset to the channel power. The actual output power at the port is equal to the channel power $P_b$ *(Channel – Stimulus – Power)* plus the *Port Power Offset.* It is equal to the *Port Power Offset* (in dBm) if *0 dBm* is selected instead of $P_b$. For power sweep and the selection $P_b$, the actual port power varies across the sweep, for other configurations the port power is constant.
- *Cal Power Offset* 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 is equal to the actual output power at the port plus n dB. The *Cal Power Offset* has no impact on the source power.

**Use of an amplifier in the signal path**

Assume that a DUT requires a constant input power of +35 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 zero dBm. With a *Port Power Offset* of +5 dB at the calibrated source port and a *Cal Power Offset* of +30 dB, the source power calibration ensures that the constant input power of +35 dBm is maintained across the entire sweep range. The actual output power of the analyzer is +5 dBm.

Note that a power calibration with an appropriate *Cal Power Offset* will automatically prevent excess input levels at the DUT.

**Remote control:**
```
SOURce<Ch>:POWer[:LEVel][:IMMediate]:OFFSet <numeric_value>, ONLY
| CPAD
SOURce<Ch>:POWer<Pt>:CORRection:LEVel:OFFSet
SOURce<Ch>:POWer<Pt>:CORRection:GENerator<Gen>:LEVel:OFFSet
```

## Receiver Power Cal...

Opens the configuration dialog for a receiver power calibration. The receiver power calibration data is acquired in a calibration sweep after appropriate selection of the calibrated wave quantity and the source.

**Purpose of a receiver power calibration**

A receiver power calibration ensures that the power readings at a specified input port of the analyzer agree with the source power level calibrated at an arbitrary reference plane. Typically, the reference plane is at the input of the receiver so that the calibration eliminates frequency response errors in the calibrated receiver.

A receiver power calibration generally improves the accuracy of power (wave quantity) measurements.

The correction data acquired in a frequency or power sweep is re-used if a *Time* or *CW Mode* sweep is activated.

**Calibration procedure and calibration sweep**

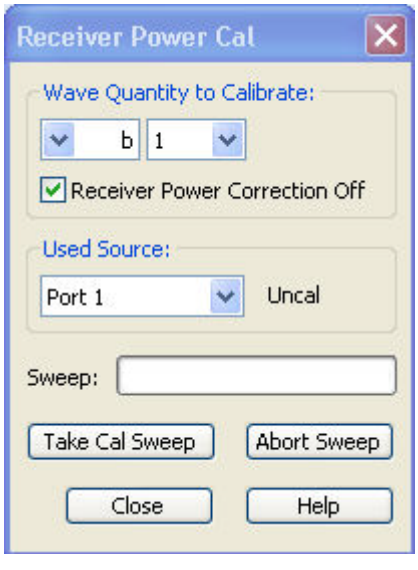
The receiver power calibration is based on a wave with known power.

- No external connection is needed for a receiver power calibration on the reference receiver (wave quantity $a_n$ with *Port n* used as a source). Besides, the reference receiver is automatically calibrated together with any source power calibration of an analyzer port unless this feature is disabled in the *Modify Source Power Cal Settings* dialog.
- Calibration of a received wave $b_n$ involves an external test setup. The received wave to calibrate can be generated by the analyzer or an external generator.

The receiver power calibration involves a single calibration sweep. The calibration sweep is performed with current channel settings but with a maximum IF bandwidth of 1 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.

An acoustic signal indicates the end of the calibration sweep. At the same time, a message next to the calibrated source indicates the status of the receiver power calibration. After a change of the sweep points or sweep range, the analyzer interpolates or extrapolates the calibration data.



The *Receiver Power Cal* dialog provides the following control elements:

- *Wave Quantity to Calibrate* provides two pull-down lists to select one of the transmitted waves $a_n$ or one of the received waves $b_n$ for the calibration. Each power calibration only applies to the selected wave quantity in the active measurement.
- *Receiver Power Correction Off* can be selected/deselected as soon as power calibration data has been acquired for the active wave quantity. If the box is checked, the calibration data is kept in memory but not applied to the wave quantity. The *Power Correction Off* softkey disables the power correction for all receivers and sources.
- *Used Source* contains all analyzer ports or external generators which can which can be used as a source for the calibrated wave quantity. The list contains all analyzer ports *Port 1* to *Port n*. Generators *(Gen 1, Gen 2, ...)* must be configured explicitly in the *System Configuration – External Generators* dialog before they appear in the list. Behind the source, a message indicates the status of the power calibration.
- *Take Cal Sweep* initiates a calibration sweep for the selected wave quantity and source. The progress of the sweep is monitored in the *Sweep* progress bar.
- *Abort Sweep* stops a running sweep. The data acquired in the aborted sweep is discarded.

*The accuracy of the receiver power calibration depends on the accuracy of the source. Depending on the test setup and reference plane, it is recommendable to perform a source power calibration before the receiver power calibration.*

**Remote control:**
```
[SENSe<Ch>:]CORRection:POWer:ACQuire
[SENSe<Ch>:]CORRection:POWer:AWAVe[:STATe]
[SENSe<Ch>:]CORRection:POWer:DATA
[SENSe<Ch>:]CORRection:POWer[:STATe]
```

## Status Messages for Power Calibration

The status of a power calibration is indicated in the trace list and in the *Used Source* panel of the *Receiver Power Cal* dialog. The following messages can be displayed:

- *PCal* denotes an active power calibration with unchanged sweep points (no interpolation or extrapolation).
- *PCai* denotes an active, interpolated power calibration (at least one calibration step is interpolated).
- *PCa?* denotes an active power calibration with changed step attenuator setting and disabled ALC.
- *PCao* denotes an active power calibration with a changed source power (in at least one calibration step).
- *PCax* denotes an active, extrapolated power calibration (in at least one calibration step).
- *Uncal* denotes an uncalibrated source (not available for traces).

The labels appear in the trace list, provided that the measured quantity is an S-Parameter, wave quantity, ratio, or a derived quantity (stability factor, PAE), and that the necessary calibrations have been performed.

**Necessary Calibrations**

A trace is marked as power calibrated as soon as the analyzer assumes that the effects of an inaccurate source or receiver power have been consistently removed. The conditions for consistency depend on the measured quantity (see also *Data Flow)*:

- S-parameters $S_{xy}$ represent the ratio of a received wave $b_x$ to a driving wave $a_y$. The system

error correction for S-parameters generally includes a receiver power calibration, however, a source power calibration can still improve the accuracy. Therefore S-parameters are assumed to be power calibrated as soon as the driving wave has been power calibrated. The same holds for quantities derived from S-parameters, e.g. stability factors.

- For wave quantities, ratios, and derived quantities (e.g. the PAE), the driving waves and all receivers must be power calibrated.

## 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.

## Power Correction On

Applies all available source and/or receiver power calibrations to the active channel. This function is disabled if no power correction data has been acquired for the active channel.

Uncheck the *...Power Correction Off* boxes in the *Source Power Cal* and *Receiver Power Cal* dialogs to enable an individual source or receiver calibration. Press *Power Correction Off* to disable the power calibration of all sources and receivers.

**Remote control:** `SOURce<Ch>:POWer:CORRection:STATe ON`
`[SENSe<Ch>:]CORRection:POWer:STATe ON`

## Power Correction Off

Disables all source and receiver power calibrations of the active channel so that all power correction data is ignored. This function is disabled if no power correction data has been acquired for the active channel.

Check the *...Power Correction Off* boxes in the *Source Power Cal* and *Receiver Power Cal* dialogs to disable an individual source or receiver calibration. Press *Power Correction On* to re-enable the power calibration of all sources and receivers.

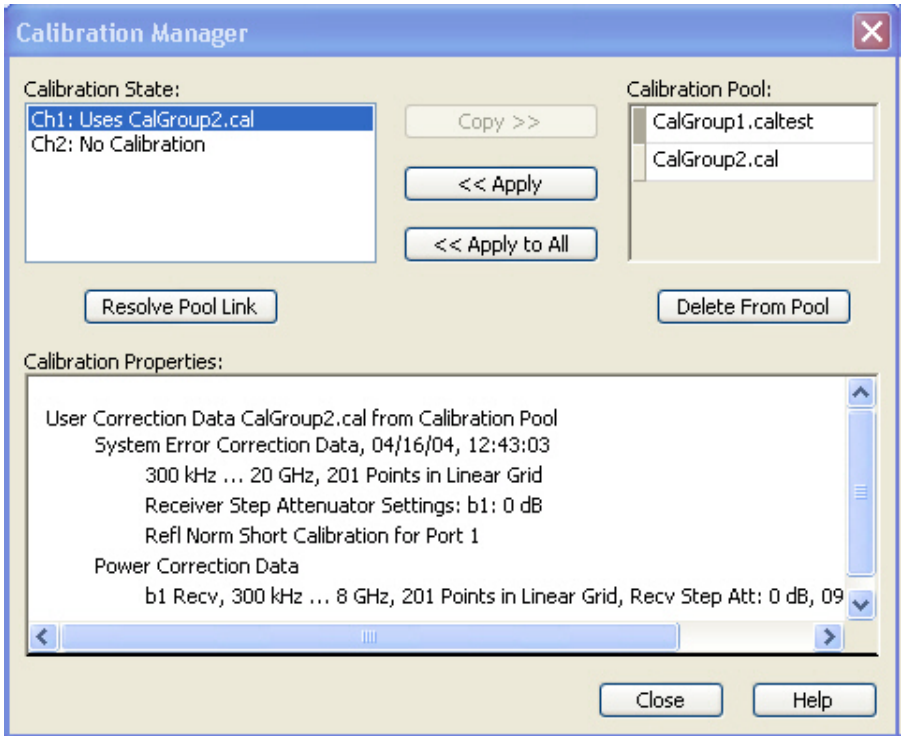
**Remote control:** `SOURce<Ch>:POWer:CORRection:STATe OFF`
`[SENSe<Ch>:]CORRection:POWer:STATe OFF`

## Calibration Manager

Opens a dialog to store system error correction and power correction data to the cal pool and to assign stored correction data to channels.

**Cal pool**

The cal pool is a collection of correction data sets (cal groups) that the analyzer stores in a common directory *C:\Rohde&Schwarz\NWA\Calibration\Data.* Cal groups in the pool can be applied to different channels and setups. 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.




The *Calibration Manager* dialog contains three main panels:

- *Calibration State* lists all channels and their current calibration state.
- *Calibration Pool* contains a list of all stored correction data sets.
- *Calibration Properties* displays the basic channel settings, the *System Error Correction Data,* and the *Power Correction Data* which are available for the channel selection in the *Calibration State* panel.

**Calibration states**

One of the following calibration states is assigned to each of the channels listed in the *Cal State* table:

- No Calibration No specific channel correction data is available. The factory system error correction is used. This is the default situation when a new setup is created *(File – New)*.
- Channel Cal A system error correction and/or power correction with channel-specific correction data is active. A new calibration of the channel affects the channel-specific correction data only.
- Uses <CalGroup> A system error correction with one of the correction data sets (cal groups) stored in the cal pool is active. A new calibration of the channel replaces the correction data in the cal pool and may affect several channels, see below.

The channel calibration of the active channel is also applied to a new channel created via *Add Channel + Trace.Corr Off* indicates that the system error correction for the channel has been deactivated using *Correction Off.*

*If a new calibration is performed for a channel assigned to a cal group (marked as Uses <CalGroup>), the correction data overwrites the cal group data, so the new calibration will affect all channels assigned to the cal group. The network analyzer generates a notice message "New calibration will overwrite cal pool!" when opening the first dialog (Select Physical Port Connectors) of the calibration wizard.*

The buttons in the upper half of the dialog are used to modify the Cal Pool and apply pool data to channels:

- *Copy >>* copies the correction data *(Channel Cal)* of the selected channel to the cal pool, generating a new pool member (cal group).
- *Apply* assigns the selected cal group to the selected channel.
- *Apply All* assigns the selected cal group to all channels in the *Cal State* table.
- *Resolve Pool Link* deletes the assignment between the selected channel *(Ch<n>: Uses <CalGroup_name>)* and the cal group. The correction data from the cal group *<CalGroup_name>* is used as a channel calibration for *<Ch1>*, the *Calibration State* list displays *Ch<n>: Channel Cal.*
- *Delete From Pool* deletes the selected cal group.

*You can apply any correction data set (cal group) from the Cal Pool to several channels, which may belong to different setups. When doing make sure that the port configuration of the channels (connectors, state, ...) and the cal group are compatible.*
*You can also use a cal group stored in the Cal Pool to define a channel-specific calibration. Apply the cal group to the channel and click Resolve Pool Link.*

**Remote control:**

```
MMEMory:STORe:CORRection <Ch>, "<file_name>"
MMEMory:LOAD:CORRection <Ch>, "<file_name>"
MMEMory:LOAD:CORRection:RESolve <Ch>, "<file_name>"
MMEMory:DELete
"C:\Rohde&Schwarz\NWA\Calibration\Data\<file_name>"
MMEMory:DELete:CORRection "<file_name>"
[SENSe<Ch>:]CORRection:DATE?
[SENSe<Ch>:]CORRection:DATA:PARameter?
[SENSe<Ch>:]CORRection:SSTate?
```

## Calibration Kits

Opens a dialog to manage the calibration kits in use, add new kits and import or export kits.

**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.

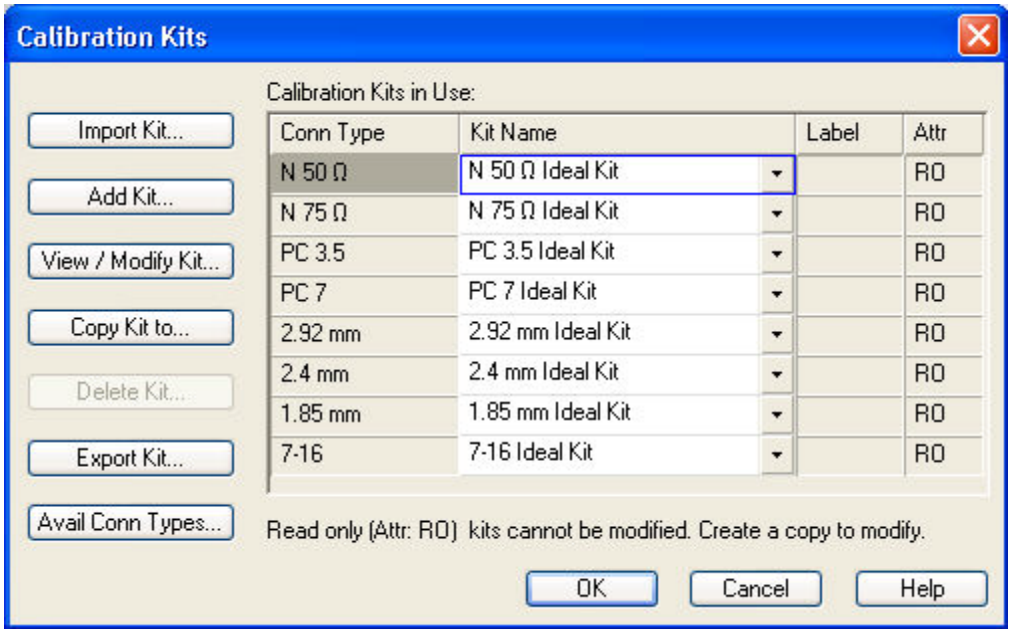
The standards are grouped into several types (open, through, match,...) corresponding to the different input quantities for the analyzer's error models. The standard type also determines the equivalent circuit model used to describe its properties. The circuit model depends on several parameters that 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 by means of S-parameter tables stored in a file.

The analyzer provides a large number of predefined cal kits but can also import cal kit files and create new kits:

- A selection of predefined kits with different types of parameters is available for all connector types. The parameters of these kits are displayed in the *Add/Modify 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 sub-dialogs.

Calibration kits and connector types are global resources; the parameters are stored independently and available irrespective of the current setup.




The *Calibration Kits* dialog contains a series of buttons, each of them opening a dialog:

- *Import Kit* imports a cal kit file containing the parameters for a new calibration kit. It is possible to load several kits, so the analyzer asks whether to set the imported calibration kit active.
- *Add Kit* combines different standards to form a new calibration kit.
- *View/ Modify Kit* adds or deletes standards in an imported or user-defined calibration kit and/or changes their parameters.
- *Copy Kit to...* creates a copy of the selected kit with a new cal kit name.
- *Delete Kit* removes an imported or user-defined kit from the list of *Calibration Kits in Use*.
- *Export Kit* exports calibration kit data to a cal kit file.
- *Avail. Conn. Types* adds or deletes connector types.

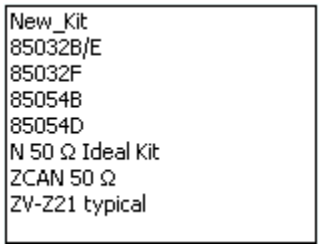
The *Calibration Kits in Use:* table is used to select a calibration kit for each connector type. The pull-down lists contain all kits for the individual connector types.

**Remote control:**

```
[SENSe<Ch>:]CORRection:CKIT:<conn_type>:SELect '<ckit_name>'
[SENSe<Ch>:]CORRection:CKIT:SELect '<conn_type>',
'<ckit_name>'
```

## 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 annexed to the kit name:




- **Universal** parameters (no label) describe calibration kit models with highly standardized components so that the parameters are valid for all calibration kits of the model.
- **Typical** parameters (labelled *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 (labelled *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 determine the accuracy of the system error correction and of the measurements. The calibration wizard 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 following standards:

| Standard (Gender) | R (Load) | Electrical Length (Offset) |
|---|---|---|
| Open (f, m) | ∞ Ω | 0 mm (Delay: 0 s) |
| Short (f, m) | 0Ω | 0 mm |
| Offset Short (f, m) | 0Ω | 10 mm |
| Match (f, m) | $Z_0$ (reference 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: $Z_0$ (reference impedance of the connector type)
- Loss: 0 dB / sqrt(GHz) (0 GΩ / s)

- All inductance and capacitance parameters are set to zero.

## Cal Kit Files

Calibration kit files can be used to store the parameters of a particular calibration kit, to reuse 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 setup 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 (.snp file) that are necessary to determine its magnitude and phase response.
- To **export** cal kit data, the analyzer uses a specific binary file format *.calkit.
- Three different **import** file formats are supported: ZVA-specific binary cal kit files (*.calkit), ZVR-specific binary cal kit files (*.ck), cal kit files in Agilent-specific ASCII formats (*.csv, *.prn; see notes below).

By default cal kit files are stored in the C:\Rohde&Schwarz\NWA\Calibration\Kits directory.

TIP To import a ZVA-specific or ZVR-specific cal kit file (*.ck or *.calkit) you can also use the Windows Explorer and simply double-click the file or drag and drop the file into the NWA application. The imported cal kit file is automatically set active.

NOTE **Importing older ZVR cal kit files**

On loading some older ZVR-specific *.ck files, e.g. the ZV-Z23 cal kit file, the ZVA generates the message *File does not comply with instrument calibration kit file format*. The files must be converted using a ZVR network analyzer equipped with a firmware version V3.52 or higer. Proceed as follows:

30. On the ZVR, press *CAL – CAL KITS – MODIFY KITS – INSTALL NEW KIT* to import the *.ck file.
31. Press *CREATE INST FILE* in the same submenu to export the *.ck file in a ZVA-compatible format.
32. Import the converted file into the ZVA.

***.csv cal kit files: VNA Cal Kit Manager 2.1**

The *VNA Cal Kit Manager* is a free, Windows-based software tool intended to import, edit, and export *csv cal kit files. The software is available for download at http://www.vnahelp.com/products.html. The decimal separator used by the *VNA Cal Kit Manager V2.1* depends on the language version of the Windows operating system: Cal kit files generated on an English operating system contain dots, the ones generated on a German system contain commas.

The network analyzer expects the dot as a separator and displays an error message when a *.csv file with commas is loaded. Please install the *VNA Cal Kit Manager V2.1* on an appropriate (e.g. Englisch) Windows version to avoid trouble.

***.prn cal kit files: PNA Cal Kit Editor**

The network analyzer can import and process cal kit files created with the *PNA Cal Kit Editor*. The files use the extension *.prn; the data format is identical to the *.csv format.

The decimal separator used by the *PNA Cal Kit Editor* depends on the language version of the Windows operating system: Cal kit files generated on an English operating system contain dots, the ones generated on a German system contain commas.

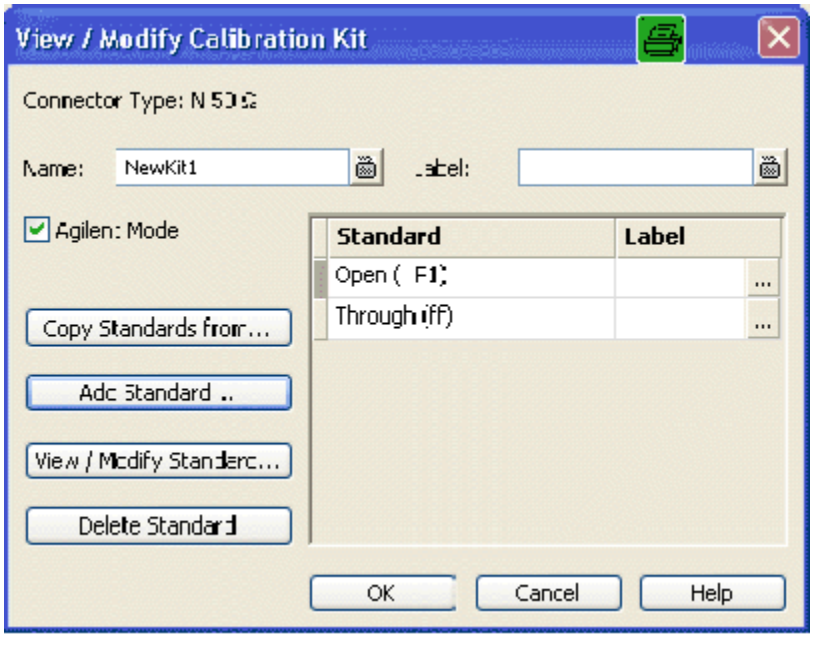
The network analyzer expects the dot as a separator and displays an error message when a *.prn file with commas is loaded. Please install the *PNA Cal Kit Editor* on an appropriate (e.g. Englisch) Windows version to avoid trouble.

**Remote control:**
```
MMEMory:LOAD:CKIT "file_name"
MMEMory:STORe:CKIT "kit_name", "file_name"
[SENSe<Ch>:]CORRection:CKIT:INSTall "<file_name>"
```

## Add / View / Modify Calibration Kit

Combines different standards to form a new calibration kit, assigning a connector type and further attributes (name, labels). This dialog is opened from the *Calibration Kits* dialog using the *Add Kit...* button (the dialog title is *Add Calibration Kit)* or using the *View/Modify Kit* button (the dialog title is *View Calibration Kit* or *View / Modify Calibration Kit).* The functionality of both dialog versions is the same.

*If Add Calibration Kit is used for a predefined calibration kit, the View / Modify Standard... button can be used to open the View / Modify Standard in <kit_name> dialog check the standard parameters. All other controls are inactive.*




In its upper part the *Add Calibration Kit* dialog contains several controls to do the following:

- Select a predefined or user-defined *Connector Type.*
- Insert the (unique) *Name* of the new kit and assign a *Label*.
- Qualify whether or not the *Offset* and *Load* parameters for the standards are defined and displayed in *Agilent Mode*.

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.

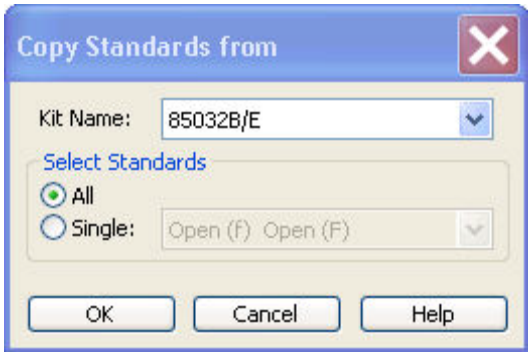
The table in the center of the dialog lists all standards in the new kit together with either their gender or their port assignment (see *Restrict Port Assignment)* and the label (if defined). Clicking a standard opens the *View / Modify Standard in <kit_name>* dialog where it is possible to change the parameters. Four buttons can be used to **change the entries in the list.**

- *Copy Standards from...* opens a dialog to copy standards from another calibration kit to the new kit.
- *Add Standard...* selects, defines and adds a new standard to the kit. A calibration kit may only contain a single standard of each type.
- *View / Modify Standard...* displays or changes the properties of the selected standard.
- Delete Standard removes the selected standard from the list and from the calibration kit.

Use *Copy Standards from...* to avoid the (re-)definition of their properties. Standards are copied together with their parameters and their label, which you can modify once they belong to the new kit.

## Copy Standards from

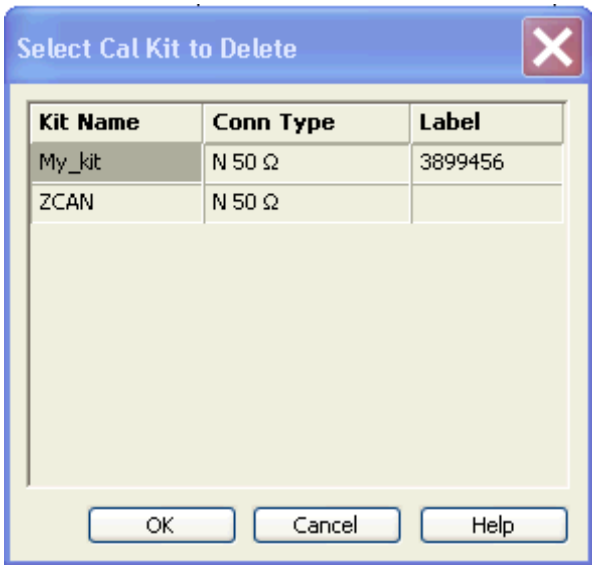
Copies standards together with their parameters from a calibration kit in use to a new calibration kit. This dialog is opened from the *View / Modify Calibration Kit* dialog *(Copy Standards from...* button).




In the dialog, it is possible to select one of the calibration kits in use *(Kit Name)* and select either all or a single standard to be copied. A calibration kit may only contain a single standard of each type.

## Select Cal Kit to Delete

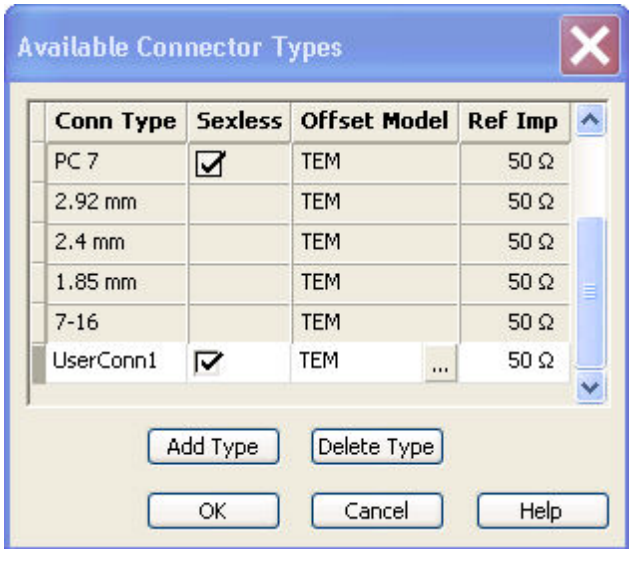
Displays a list of calibration kits to be deleted. This dialog is opened from the *Calibration Kits* dialog *(Delete Kits...* button).




The table shows the imported and user-defined calibration kits with their name, connector type and label (if defined). *OK* deletes the selected kit. Predefined kits cannot be deleted.

## Available Connector Types

Displays and modifies the list of available connector types. This dialog is opened from the *Calibration Kits* dialog *(Connector Types...* button).




The list shows the available connector types with their name *(Conn Type)*, polarity *(Sexless)* and reference impedance *(Ref Imp)*. The *Offset Model* describes the mode of wave propagation in the transmission lines of the standards associated with the connector type. The buttons below are used to add and delete user-defined connector types. Deleting a connector type will also delete all calibration or adapter kits assigned to it.

**Impact of reference impedance**

The reference impedance *(Ref Imp)* $Z_0$ for the connectors is a critical value that is displayed in the *Port Configuration* menu and represents an input value for various parameter conversions. $Z_0$ enters into:

- The calculation of the S-parameters for the calibration standards associated with the connector type, provided that they are derived from a circuit model *(Add/Modify Standard* dialog).
- The calculation of the (default) reference impedances for balanced ports *(Def. Balanced Port* tab of the *Balanced and Measured Ports* dialog).
- The calculation of impedance and admittance parameters.

**Storing connector type settings**

Calibration kits and connector types are global resources; the parameters are stored independently and available irrespective of the current setup. The connector type settings are always stored together with the associated calibration kit parameters. It is possible to export and import the connector settings using *Export Kit...* and *Import Kit...* in the *Calibration Kits* dialog.

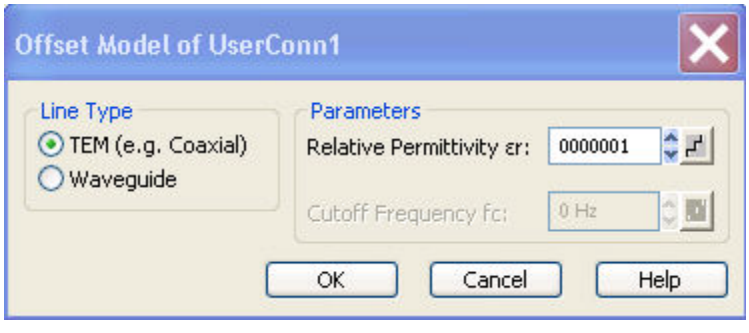
The name, polarity and reference impedance of a user-defined connector can be changed in the table. The ... button in the *Offset Model* column opens the *Offset Model* dialog to define the propagation mode in detail.

*After assigning a calibration or adapter kit to a user-defined connector type, you can still change its name, offset model and reference impedance. Switching between sexed and sexless will delete all kits assigned to the connector type.*

**Remote control:** `[SENSe<Ch>:]CORRection:CONNection`
`[SENSe<Ch>:]CORRection:CONNection:DELete`

## Offset Model of <Standard>

Defines the mode of wave propagation in the lines of the standards associated with the connector type. This dialog is opened from the *Available Connector Types* dialog ( ... button in the table).




The *Parameters* to be selected depend on the *Line Type:*

- If the calibration kit standards contain lines with transverse electric propagation mode (TEM), then the *Relative Permittivity* $\varepsilon_r$ of the dielectric can be defined. The default permittivity is the value for air. TEM-type lines have not cutoff frequency.
- If the calibration kit standards contain waveguides, then the lowest frequency where a wave propagation is possible *(Cutoff Frequency* $f_c$*)* can be defined. The default cutoff frequency if 0 Hz (propagation at all frequencies). No relative permittivity is needed for waveguides.

*The impedance for waveguides is frequency-dependent. If a waveguide line type is selected, various dialogs (e.g. Add Standard...) will indicate varies instead of a definite impedance value.*

**Impact of offset model parameters**

The offset model parameters are used for the calculation of the S-parameters for the calibration standards associated with the connector type, provided that they are derived from a circuit model *(Add/Modify Standard* dialog).

- For TEM-type lines, the relative permittivity $\varepsilon_r$ is needed for the conversion of and ZVR-type *Loss* (in units of dB/sqrt(GHz)) into an Agilent-type *Offset Loss* (in units of GΩ/s) and vice versa (see *Offset* and*Load* parameters). The *Electrical Length* and *Delay* values in the *Modify Offset* dialog are directly entered and therefore independent of $\varepsilon_r$.
- 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 in order to acquire calibration data, the analyzer checks the low frequency cutoff. If the start frequency of the sweep range is below $f_c$ , then the calibration wizard generates and error message.

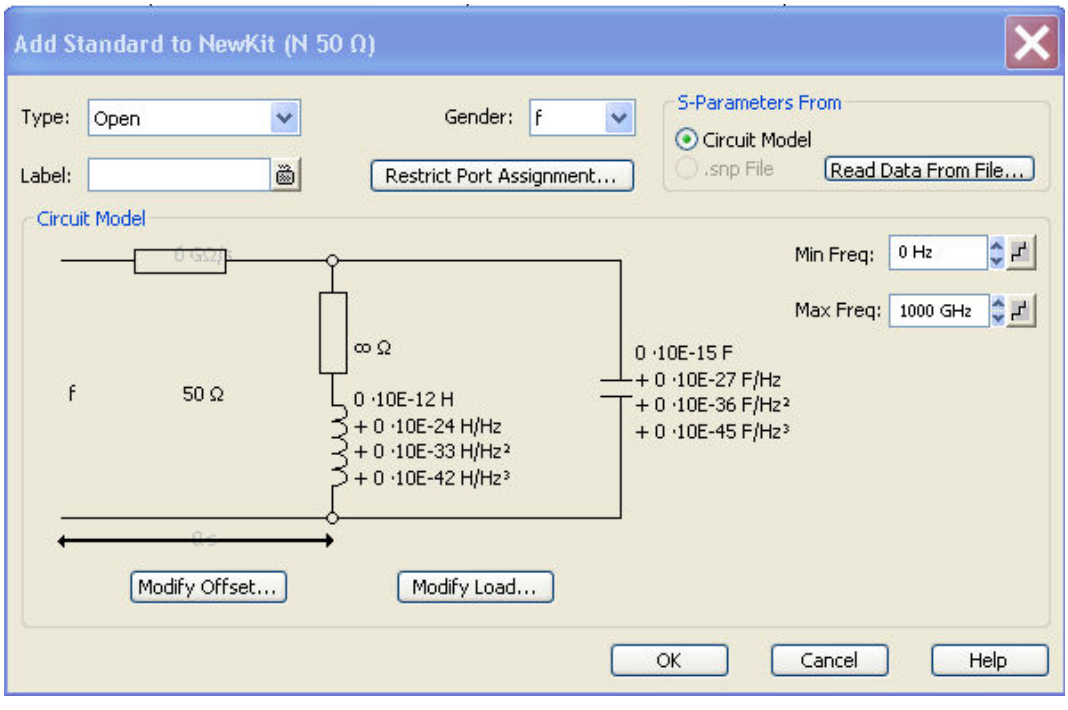
The offset model parameters are not used except in the context of calibration. The offset parameterdefinitions (see*Mechanical Length)* are based on independent $\varepsilon_r$values.

**Remote control:** `[SENSe<Ch>:]CORRection:CONNection`

## Add or View / Modify Standard

Defines, displays or modifies the properties of the calibration standards in a particular calibration kit. This dialog is opened from the *Add or View / Modify Calibration Kit* dialog *(Add Standard...* or *View / Modify Standard...* buttons). Depending on the title, some control elements may not be active.

For an overview of calibration standards and their properties refer to section Calibration standard types below.



In its upper part the *Add Standard* or *View / Modify Standard* dialog contains several controls to do the following:

- Select a standard *Type* and its *Gender* (for polarized/not sexless connector types and if the port assignment is not restricted) and assign a *Label*.
- *Restrict Port Assignment*
- ***Select S-Params From***

  Qualify whether the standard is described by a *Circuit Model* from which the analyzer can calculate the S-parameters or by a table of measured or simulated S-parameters stored in a Touchstone file. Pressing the *Read Data from File...* button opens a file selection dialog where the appropriate file type (*.s1p for one-port standards and *.s2p for two-port standards) is automatically selected.

  NOTE The *Sliding Match* and *Attenuation* are special standard types which must be described by a circuit model. The controls in the *S-Params From* panel are disabled.

  For two-port standards described by a *.s2p file, the implicit ports 1 and 2 (given by the order of S-parameters Re(S11) Im(S11) Re(S21) Im(S21) Re(S12) Im(S12) Re(S22) Im(S22) in the file) are assigned to the test ports that the analyzer actually calibrates as follows: Port 1 is always assigned to the lower-numbered calibrated test port, port 2 to the other (higher-numbered) calibrated test port.

TIP Assigning a label to standards is optional. However, the label is displayed in many dialogs and can provide useful information about the standard, e.g. its serial number.

If *Circuit Model* is selected in the *S-params From* panel, then the controls in the central panel of the dialog are enabled. The circuit diagram is adjusted to the selected standard type. The following parameters can be set:

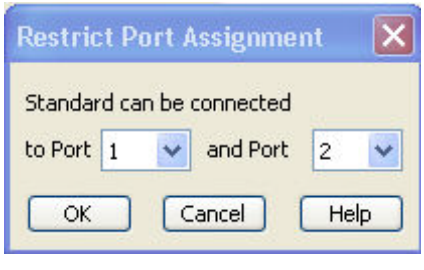
- Frequency range *(Min. Freq.* to *Max. Freq)* for which the circuit model is valid. During calibration, the analyzer checks whether the sweep range is contained in the validity range of all measured standards and possibly generates a warning (see *Measure Standards* dialog).
- *Offset* and *Load* parameters of the circuit model.

NOTE *The impedance for waveguides is frequency-dependent. If a waveguide line type is selected in the Offset Model dialog, the circuit model indicates varies instead of a definite impedance value.*

**Remote control:**
```
[SENSe<Ch>:]CORRection:CKIT:<std_type>
[SENSe<Ch>:]CORRection:CKIT:<conn_type>:<std_type>
MMEMory:LOAD:CKIT:SDATa
```

## Restrict Port Assignment

Opens a dialog to define whether the standard can be connected to any port of the analyzer or to just one port (for one-port standards) or a pair of ports (for two-port standards).



The port assignment is displayed in the *Add or View / Modify Calibration Kit* dialogs.

**Port assignment and gender**

The standards are handled differently, depending on their port assignment:

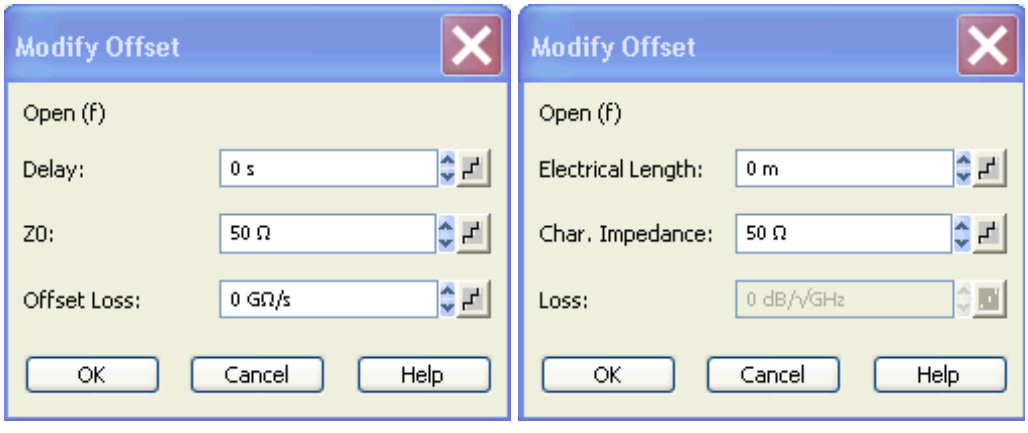
- If the port assignment is not restricted, the gender belongs to the definition of polarized standards. When the connector type and calibration kit is selected in the calibration wizard, the analyzer checks whether the kit contains the necessary standard types and whether the standards have the right gender.
- Standards with restricted port assignment are assumed to have the right gender (the one required for this port). In the *View Modify Standard* dialog, the *Gender:* input field is disabled. In the calibration wizard, the analyzer checks whether the kit contains the necessary standard types for the required ports. Instead of the gender, the port assignment is stored in the calibration kit file.

This approach simplifies the definition of standards and helps to avoid inconsistencies.

**Remote control:** `[SENSe<Ch>:]CORRection:CKIT:<std_type>`

## Modify Offset

Specifies the offset parameters for the transmission lines of a particular calibration standard. This dialog is opened from the *Add or View / Modify Standard...* dialog (*Modify Offset...* button).




The offset parameters depend on whether or not the circuit model is defined in *Agilent Mode* (see *Add/Modify Calibration Kit* dialog):

- If *Agilent Mode* is active, then the standard is characterized by its *Delay* (in s), its characteristic impedance $Z_0$ (in Ω) and its *Offset Loss* (in GΩ/s).
- If *Agilent Mode* is switched off, then the standard is characterized by the ZVR-compatible parameters *Electrical Length* (in m), its *Char. Impedance* (in Ω) and its *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*; $Z_0$ corresponds to the *Char. Impedance.* Moreover the analyzer converts an Agilent-type *Offset Loss* into a ZVR-type *Loss* and vice versa using the *Relative Permittivity* $\varepsilon_r$ for the connector type defined in the *Offset Model...* dialog.

**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 and is a measure for the length of transmission line between the standard and the actual calibration plane. For a waveguide with permittivity $\varepsilon_r$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 one-port and 2-port standards.

- $Z_0$ is the *Characteristic Impedance* of thestandard. 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 Agilent mode the *Offset Loss* is expressed in units of Ω/s at a frequency of 1 GHz. The following formula holds:

$$Offset\ Loss / [\Omega / s] = \frac{Loss / [dB] \cdot Z_0 / [\Omega]}{4.3429 / [dB] \cdot delay / [s]}$$

  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.

NOTE *The impedance for waveguides is frequency-dependent. If a waveguide line type is selected in the Offset Model dialog, the Char. Impedance 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.*

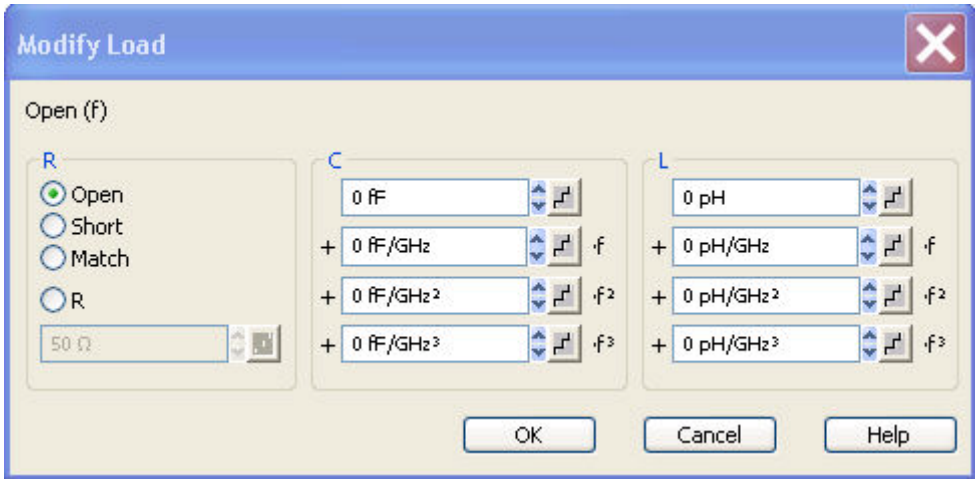
- The *Sliding Match* is a one-port standard with variable load parameters (sliding load) and unspecified length. The reference impedance is fixed and equal to the characteristic impedance of the connector type. No load and offset parameters need to be set.
- The *Attenuation* is a two-port standard which is fully matched in both directions (the reflection factor at both ports is zero). No load and offset parameters need to be set.

**Remote control:** `[SENSe<Ch>:]CORRection:CKIT:<conn_type>:<std_type>`

## Modify Load

Specifies the load parameters for a particular calibration standard describing its terminal impedance.

This dialog is opened from the *Add or View / Modify Standard...* dialog (*Modify Load...* button).




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 a special value *(Open* for ∞ Ω so that the inductance coefficients are irrelevant, *Short* for 0 Ω, *Match* for the reference impedance of the current connector type) or set 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.

**Load parameters and standard types**

Load parameters are used to describe all types of standards except a *Through,* a *Sliding Match,* aLineand 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.

**Remote control:** `[SENSe<Ch>:]CORRection:CKIT:<conn_type>:<std_type>`

## Calibration Standard Types

The following table gives an overview of the different standards and their offset and load models:

| 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) | $Z_0$ (reference 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(s) 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) | – | ☑ | ☑ |

The calibration of rectangular waveguides at very high frequencies requires a special set of standards; see *Frequency Extension – Calibration*.

**Remote control:** For an overview of standard parameters see also
`[SENSe<Ch>:]CORRection:CKIT:<conn_type>:<std_type>`

## Offset

The *Offset* menu defines a length offset and a loss for the test ports. The offset parameters complement the system error correction, compensating for the known length and loss of a (non-dispersive and perfectly matched) transmission line between the calibrated reference plane and the DUT.

**Length offset parameters: Definition**

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 and is a measure for the length of transmission line between the standard and the actual calibration plane. For a line with permittivity $\varepsilon_r$ and mechanical length $L_{mech}$ the delay and the electrical length are calculated as follows:

$$\text{Delay} = \frac{L_{mech} \cdot \sqrt{\varepsilon_r}}{c}; \quad \text{Electrical Length} = L_{mech} \cdot \sqrt{\varepsilon_r}$$

*Electrical Length, Mechanical Length* or *Delay* are coupled parameters. When one of them is changed, the other two follow.

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 with respect to the frequency (see mathematical relations). The length offset parameters compensate for a constant delay, which is equivalent to a linear phase response.

If a dispersive connector type (i.e. a waveguide; see *Offset Model* dialog) is assigned to a test port that is related to a particular quantity, then the phase of the quantity is calculated taking dispersion effects into account.

**Loss parameters: Definition**

The loss *L* is the attenuation of a wave traveling through the offset transmission line. In logarithmic representation, the loss can be modeled as the sum of a constant and a frequency-dependent part. The frequency dependence is essentially due to the skin effect; the total loss can be approximated by an expression of the following form:

$$Loss(f) = \left[Loss(f_{ref}) - Loss_{DC}\right] \cdot \sqrt{\frac{f}{f_{ref}}} + Loss_{DC}$$

The DC loss $Loss_{DC}$, the reference frequency $f_{ref}$, and the loss at the reference frequency $Loss(f_{ref})$ are empirical parameters for the transmission lines connected to each port which can be entered into any of the dialogs in the *Offset* menu. For a lossless transmission line, both $Loss_{DC}$ and $Loss(f_{ref})$ are zero. In practice, $Loss(f_{ref})$ often represents the dominant contribution so that $Loss_{DC}$ can be set to zero.

**Offset parameters: Application and effect**

Offset 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 from 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 into a (perfectly matched) transmission line at that port.

In contrast to the embedding/deembedding procedure, the offset 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.

NOTE 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 ($\lambda/4$), 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:

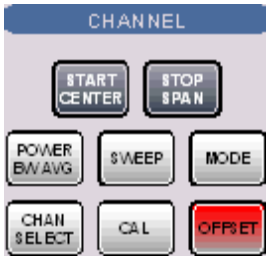
- A positive offset parameter causes a positive phase shift of the measured parameter and therefore reduces the calculated group delay.
- A negative offset parameter causes a negative phase shift of the measured parameter and therefore increases the calculated group delay.

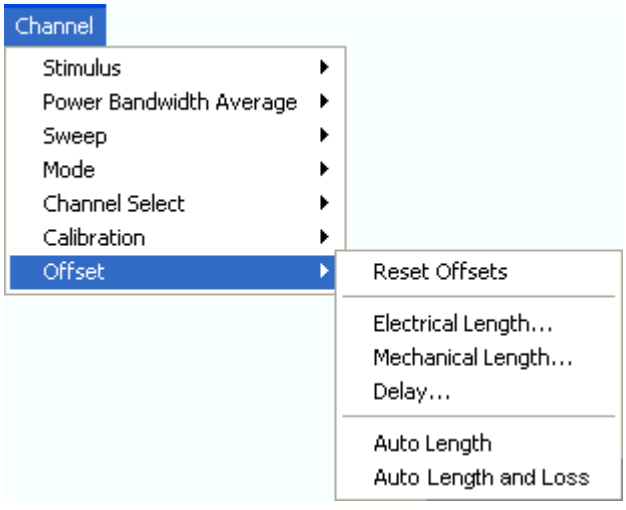
**Offset parameters for balanced ports**

The functions in the *Offset* menu can be used for balanced port configurations:

- If a balanced port configuration is active the logical and physical ports are shown in the *Electrical Length, Mechanical Length* and *Delay* dialogs.
- 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.




The offset parameters can be defined separately for each port.

- *Reset Offsets* restores the default values for all length offsets, i.e. it resets all values to zero.
- *Electrical Length* defines electrical length offsets and loss parameters at all test ports.
- *Mechanical Length* defines mechanical length offsets and loss parameters at all test ports.
- *Delay* defines delay times and loss parameters at all test ports.
- *Auto Length* determines a length offset for the receiving port of the active measured quantity with the condition that the residual group delay of the active trace is minimized.
- *Auto Length and Loss* determines a length offset an loss for the receiving port of the active measured quantity with the condition that the residual group delay of the active trace is minimized and the measured loss is minimized as far as possible across the entire sweep range.

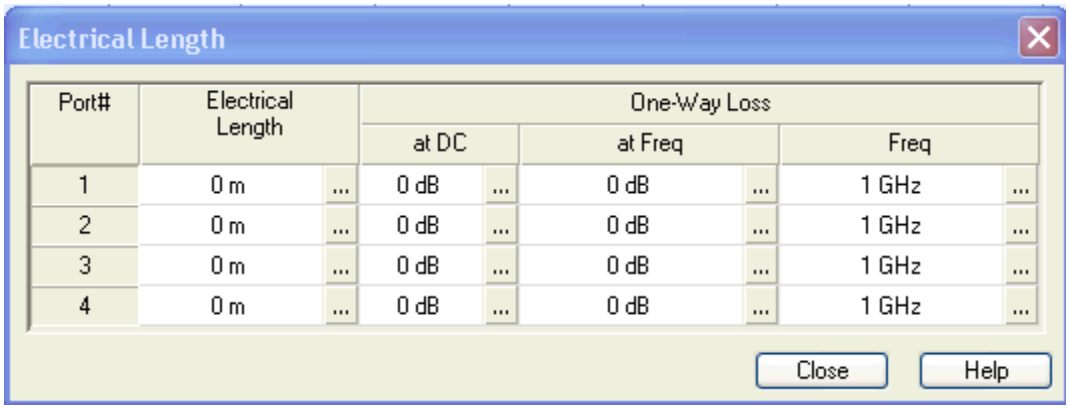
*The Zero Delay at Marker function overwrites the Offset parameters.*

**Remote control (for *Reset Offsets*):**

```
[SENSe<Ch>:]CORRection:OFFSet<port_no>[:STATe] <numeric_value>
```

## Electrical Length

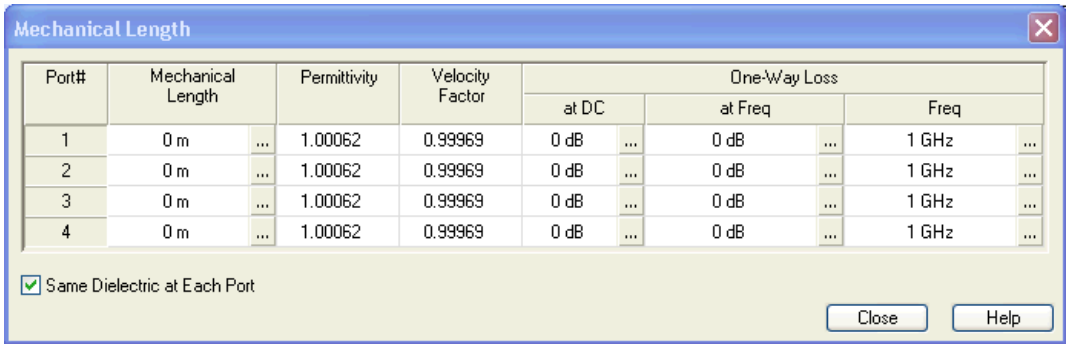
Opens a dialog to define the length offset parameters for the physical test ports as electrical lengths. The dialog also contains the DC loss, the loss at the reference frequency Loss($f_{ref}$); and the reference frequency $f_{ref}$; see *Loss parameters: Definition* above.




**Remote control:**
```
[SENSe<Ch>:]CORRection:EDELay<port_no>:ELENgth
[SENSe<Ch>:]CORRection:LOSS<port_no>
[SENSe<Ch>:]CORRection:LOSS<port_no>:FREQuency
[SENSe<Ch>:]CORRection:LOSS<port_no>:OFFSet
```

## Mechanical Length

Opens a dialog to define length offset parameters for the physical test ports as mechanical lengths and permittivities. The dialog also contains the DC loss, the loss at the reference frequency Loss($f_{ref}$); and the reference frequency $f_{ref}$; see *Loss parameters: Definition* above.



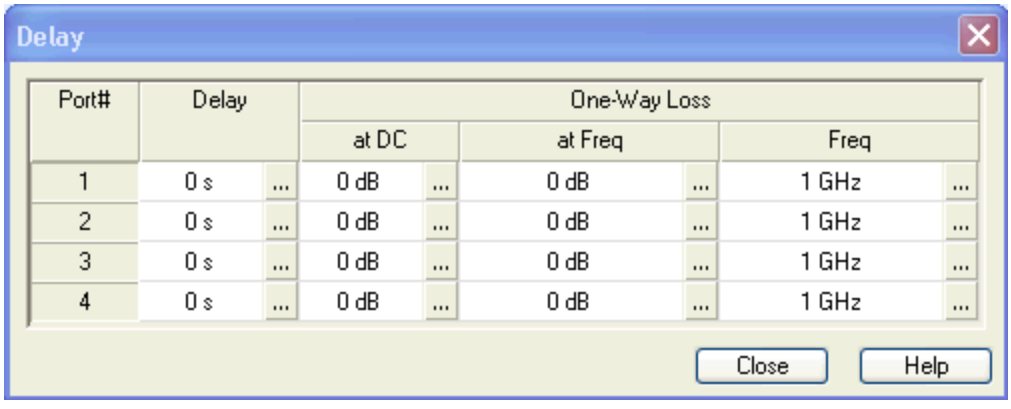

The *Mechanical Length* dialog contains the three editable columns *Mech. Length, Permittivity* ($\varepsilon_r$) and *Velocity Fact.* The velocity factor is 1/sqrt($\varepsilon_r$) and is a measure for the velocity of light in a dielectric with permittivity $\varepsilon_r$ relative to the velocity of light in the vacuum (velocity factor < 1); see *Offset parameters: Definition* above. Permittivity and velocity factor are coupled parameters.

Click *Same Dielectric at Each Port* to change the permittivity or velocity factor for all ports by entering a single value.

**Remote control:**
```
[SENSe<Ch>:]CORRection:EDELay<port_no>:DISTance
[SENSe<Ch>:]CORRection:EDELay<port_no>:DIELectric
[SENSe<Ch>:]CORRection:LOSS<port_no>
[SENSe<Ch>:]CORRection:LOSS<port_no>:FREQuency
[SENSe<Ch>:]CORRection:LOSS<port_no>:OFFSet
```

## Delay

Opens a dialog to define the length offset parameters for the physical test ports as delays. The dialog also contains the DC loss, the loss at the reference frequency Loss($f_{ref}$); and the reference frequency $f_{ref}$; see *Loss parameters: Definition* above.




**Remote control:**
```
[SENSe<Ch>:]CORRection:EDELay<port_no>[:TIME]
[SENSe<Ch>:]CORRection:LOSS<port_no>
[SENSe<Ch>:]CORRection:LOSS<port_no>:FREQuency
[SENSe<Ch>:]CORRection:LOSS<port_no>:OFFSet
```

## Auto Length

Adds an electrical length offset to the active test port with the condition that the residual delay of the active trace (defined as the negative derivative of the phase response) is minimized across the entire sweep range. If *Delay* is the selected trace format, the entire trace is shifted in vertical direction and centered around zero. In phase format, the *Auto Length* corrected trace shows the deviation from linear phase. The effect of a dispersive connector type (i.e. a waveguide; see *Offset Model* dialog) assigned to the receiving port of the measured quantity is taken into account.

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.

**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 is 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.

Use *Zero Delay at Marker* to set the delay at a special trace point to zero.

**Remote control:** `[SENSe<Ch>:]CORRection:EDELay<port_no>:AUTO ONCE`

## Auto Length and Loss

Determines all offset parameters 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 the entire sweep 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 *Auto Loss* correction modifies the magnitude of the measured quantity, leaving the (auto length-corrected) phase unchanged.

If *Auto Length and Loss* is used with a line connected to a test port, the end of the line should be left open.

**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 *Loss parameters: Definition).* 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 *Electrical Length, Mechanical Length,* and *Delay* dialogs.

**Auto Length and Loss for logical ports**

The *Auto Length and Loss* function can be used for balanced port configurations as well. If the

active test port is a logical port, then the same offset parameters are 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.

**Remote control:** `[SENSe<Ch>:]CORRection:LOSS<port_no>:AUTO ONCE`

# Display Menu

The *Display* menu provides all display settings and the functions to activate, modify and arrange different diagram areas.

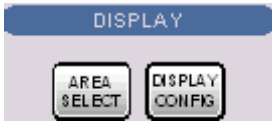
**Diagram Areas**

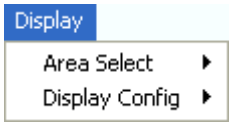
A diagram area is a rectangular portion of the screen used to display traces. Diagram areas are arranged in windows; they are independent of trace and channel settings.

- A diagram area can contain a practically unlimited number of traces, assigned to different channels (overlay mode); see Traces, Channels and Diagram Areas.
- The traces displayed in the diagram area are listed in the upper left corner.
- The channels for all traces are listed in the lower left corner.

The contents of the diagram areas are explained in section Display Elements. Diagram areas are controlled and configured by means of the functions in the *Display* menu and the following additional settings:

- The settings in the *Window* menu arrange several windows containing one or more diagram areas within the entire screen.
- Various settings to assign traces to diagram areas are provided in the *Trace – Traces* submenu.




The *Display* menu contains the following functions and submenus:

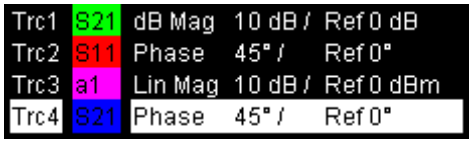
- *Area Select* provides functions to create and delete diagram areas, select an area as the active area, arrange traces to diagram areas and arrange the diagram areas in the active window.
- *Display Config* configures the entire screen and the individual diagram areas.

## Display - Area Select

The *Area Select* submenu provides functions to create and delete diagram areas, select an area as the active area, and arrange areas in the active window.

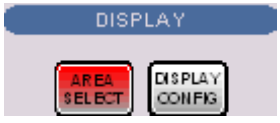
**Active and inactive traces and diagram areas**

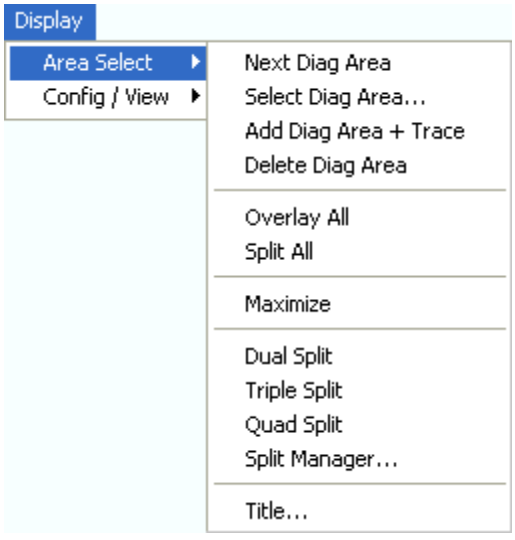
The active window can display several diagram areas simultaneously, each with a variable number of traces. One of these areas and traces is active at each time. The area number in the upper right corner of the active area is highlighted. At the same time the active trace is highlighted in the trace list on top of the active diagram area (Trc 4 in the figure below):




The analyzer provides several tools to activate a diagram area:

- A left mouse click on a point in the diagram activates the diagram including the last active trace in the diagram.
- Next Diag. Area and Previous Diag. Area activate the next/previous diagram including the last active trace in the diagram.
- A left mouse click on a trace list activates the trace including the corresponding diagram.
- Some of the functions of the *Trace – Traces* menu activate a particular trace including the corresponding diagram.




- *Next Diag. Area* selects the next diagram area as the active diagram area (disabled if only one diagram area is defined).
- *Select Diag. Area* opens a box to select an arbitrary diagram area as the active diagram area (disabled if only one diagram area is defined).
- *Add Diag Area + Trace* creates a new diagram area with a new trace.
- *Delete Diag. Area* deletes the active diagram area.
- *Overlay all* places all traces in a single diagram area which occupies the whole window.
- *Split all* splits the active window into as many diagram areas as there are traces and assigns a single trace to each area.
- *Maximize* maximizes the active diagram area to occupy the whole window.
- *Dual Split* splits the active window into two diagram areas and distributes the traces among the two areas.
- *Triple Split* splits the active window into three diagram areas and distributes the traces among the three areas.
- *Quad Split* splits the active window into four diagram areas and distributes the traces among the four areas.
- *Split Manager* opens a dialog to arrange the diagram areas in the active window.
- *Title* opens a dialog to define a title and display it in one of the diagram areas.

## Next Diag. Area

Selects the next diagram area as the active diagram area. This function is disabled if the current setup contains only one diagram area. The order of the diagram areas is given by the diagram numbers displayed in the upper right corner.

The diagram number of the active diagram area is highlighted. If one or several traces are assigned to

the next diagram area, the last active trace again becomes the active trace.

To activate any diagram area including the last active trace in this area, simply click a point inside the new area.

**Remote control:** The numeric suffix `<Wnd>` appended to the `DISPlay:WINDow<Wnd>:...` commands selects a diagram area.

## Select Diag. Area

Opens a box to select an arbitrary diagram area of the active setup as the active diagram area. This function is disabled if the current setup contains only one diagram area. The diagram area numbers are displayed in the upper right corner.

The diagram number of the active diagram area is highlighted. If one or several traces are assigned to the previous diagram area, the last active trace again becomes the active trace.

To activate any diagram area including the last active trace in this area, simply click a point inside the new area.

**Remote control:** The numeric suffix `<Wnd>` appended to the `DISPlay:WINDow<Wnd>:...` commands selects a diagram area.

## Add Diag. Area + Trace

Creates a new diagram area and a new trace, which is displayed in the new diagram area. 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.

**Remote control:** `DISPlay:WINDow<Wnd>:STATe ON`

## Delete Diag Area

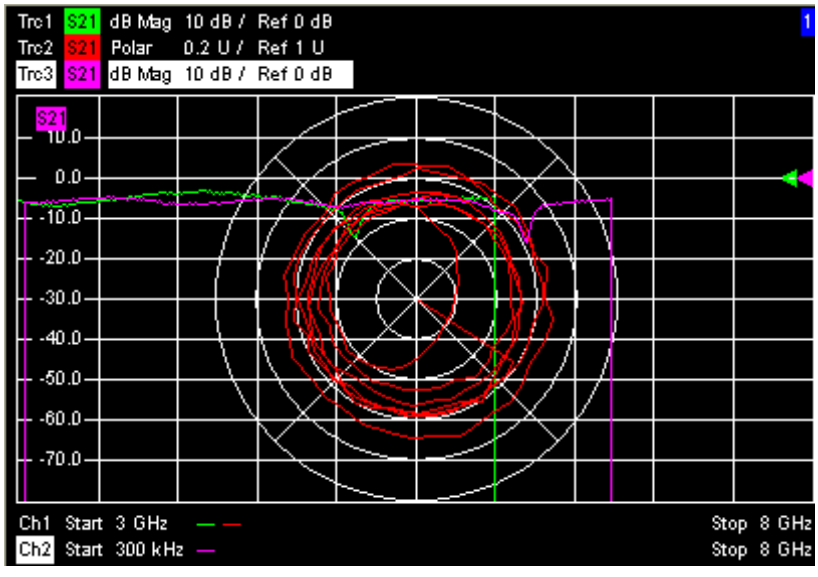
Deletes the current diagram area including all traces displayed in the diagram area. *Delete Diag Area* is disabled if the setup contains only one diagram area: In manual control, each setup must contain at least one diagram area with one channel and one trace.

To restore a diagram area that was unintentionally deleted, use the *Undo* function.

**Remote control:** `DISPlay:WINDow<Wnd>:STATe OFF`

## Overlay all

Places all traces in a single diagram area which is maximized to occupy the whole window. This function is available irrespective of the trace Format and the Channel settings; it is even possible to overlay Cartesian and polar diagrams.



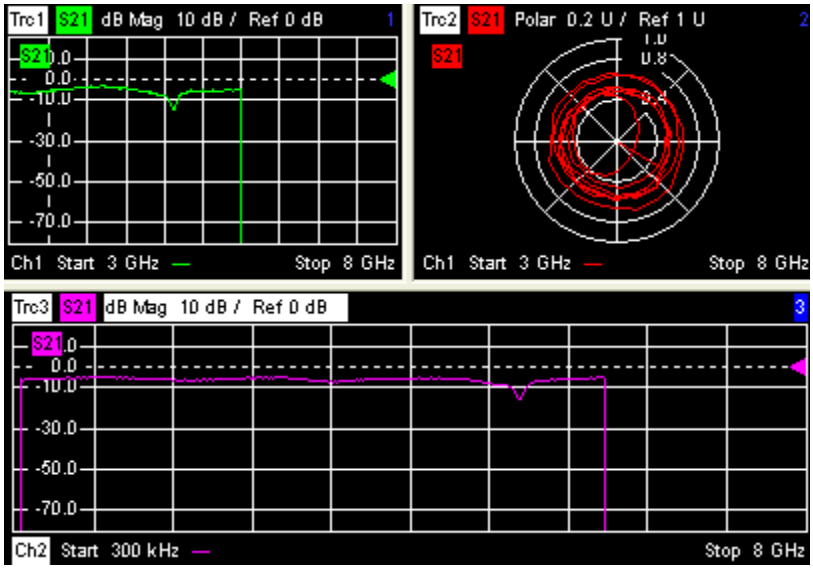

The active trace and active channel is highlighted. The scaling of the axes corresponds to the active trace.

To hide all traces except the active one, select *Split All* and *Maximize*.

**Remote control:** No command, display configuration only.

## Split all

Splits the active window into as many diagram areas as there are traces and assigns a single trace to each area.



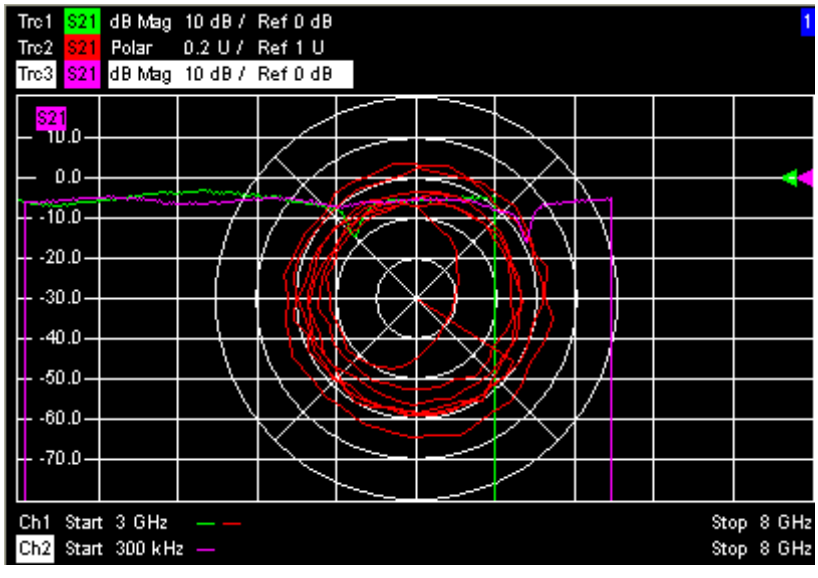

To vary the size and position of the diagram areas, drag and drop the separating frames, use the *Split Manager* or the functions of the *Window* menu.

**Remote control:** No command, display configuration only.

## Maximize

Maximizes all diagram areas of the active setup to occupy the whole window, placing the active diagram area on top. While *Maximize* is selected *Next Diag. Area* or *Select Diag. Area* can be used to activate and display the other diagram areas. The remaining commands of the *Area Select* submenu except *Title* are disabled.

Clicking *Maximize* again restores the previous display configuration.

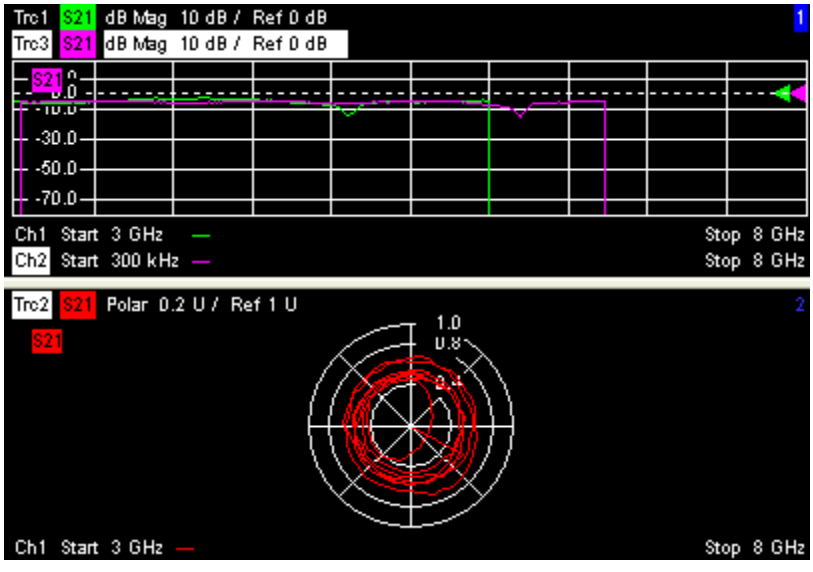


A double-click into any point in the diagram area is equivalent to the *Maximize* function. To view all traces in a common, maximized diagram area, select *Overlay all*.

**Remote control:** `DISPlay:WINDow<Wnd>:MAXimize ON | OFF`

## Dual Split

Splits the window horizontally into two diagram areas and distributes the traces among the two areas, separating diagrams with different trace Format and Channel settings (e.g. Cartesian and polar diagrams).



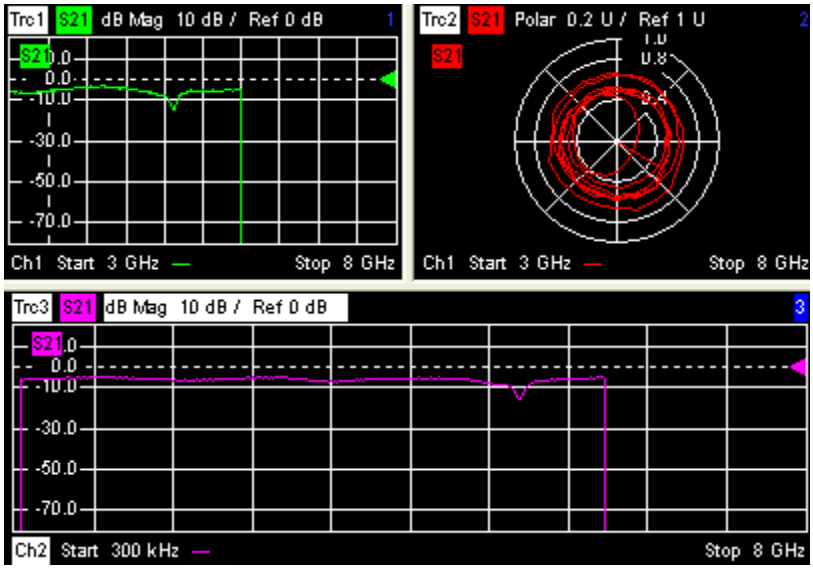

If only one trace is available, the lower diagram area is created with a new trace.

To vary the size and position of the two diagram areas, drag and drop the separating frames, use the *Split Manager* or the functions of the *Window* menu.

**Remote control:** No command, display configuration only.

## Triple Split

Splits the active window into three diagram areas and distributes the traces among the three areas, separating diagrams with different trace Format and Channel settings (e.g. Cartesian and polar diagrams).



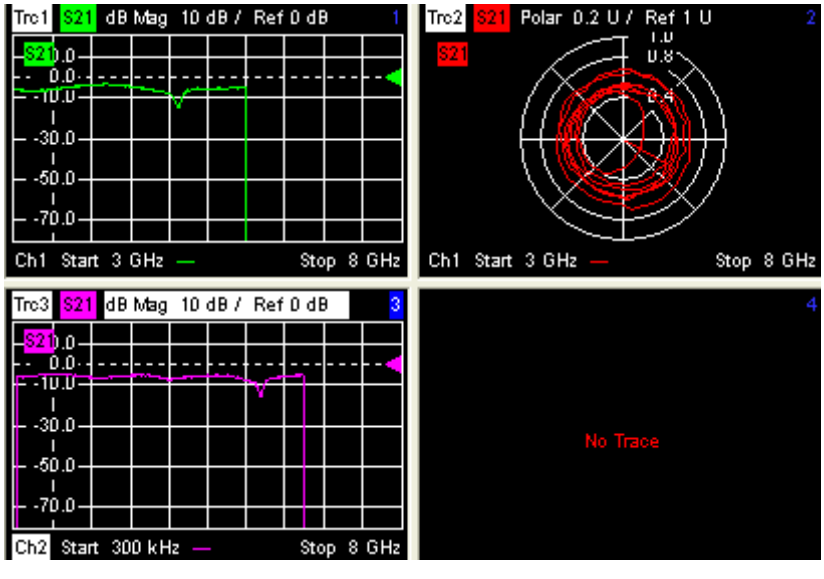

If less than three traces are available, some diagram areas are empty and display *No Trace*.

To vary the size and position of the diagram areas, drag and drop the separating frames, use the *Split Manager* or the functions of the *Window* menu.

**Remote control:** No command, display configuration only.

## Quad Split

Splits the active window into four diagram areas and distributes the traces among the four areas, separating diagrams with different trace Format and the Channel settings (e.g. Cartesian and polar diagrams).




If less than four traces are available, some diagram areas are empty and display *No Trace*.

To vary the size and position of the diagram areas, drag and drop the separating frames, use the *Split Manager* or the functions of the *Window* menu.

**Remote control:** No command, display configuration only.

## Split Manager

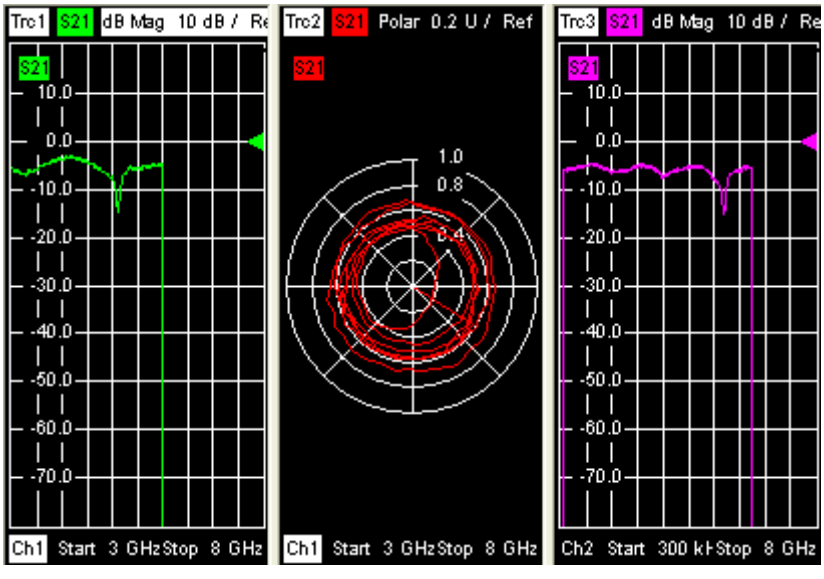
Opens a dialog to arrange the diagram areas in the active window.

- *Split Mode* provides a drow-down list to select alternative display schemes for the diagram areas (see examples for split modes below).
- *Number of Diagram Areas* indicates the total number of diagram areas. Increasing/decreasing the number creates new diagram areas or deletes diagram areas.

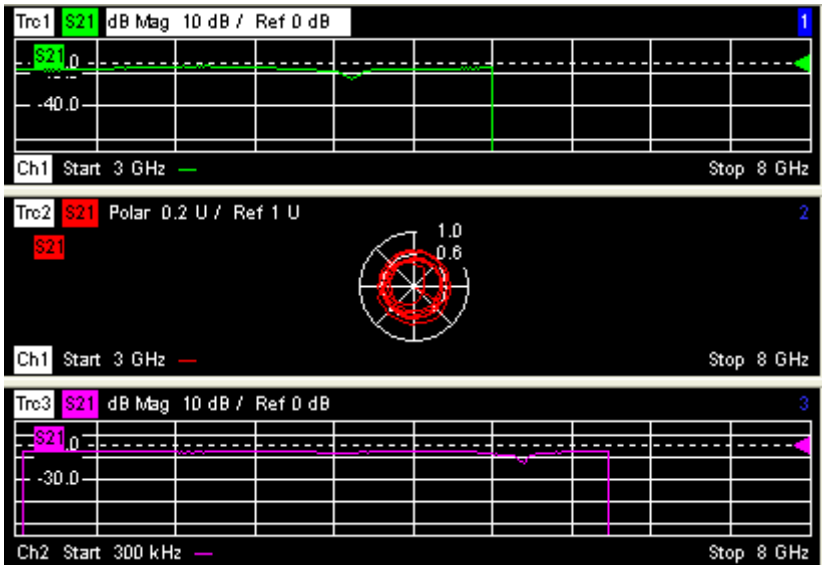
**Examples for split modes**

The following examples were obtained with 3 diagram areas, each with 1 trace.

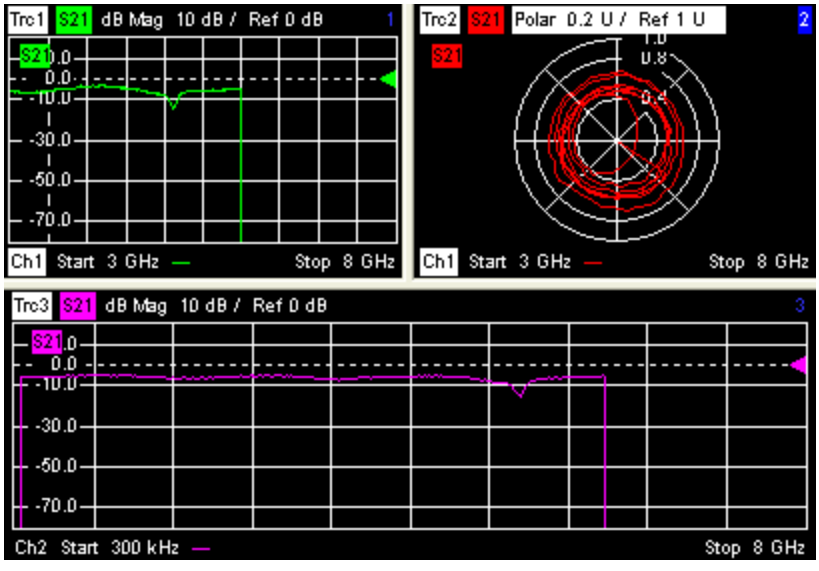
**Lineup**



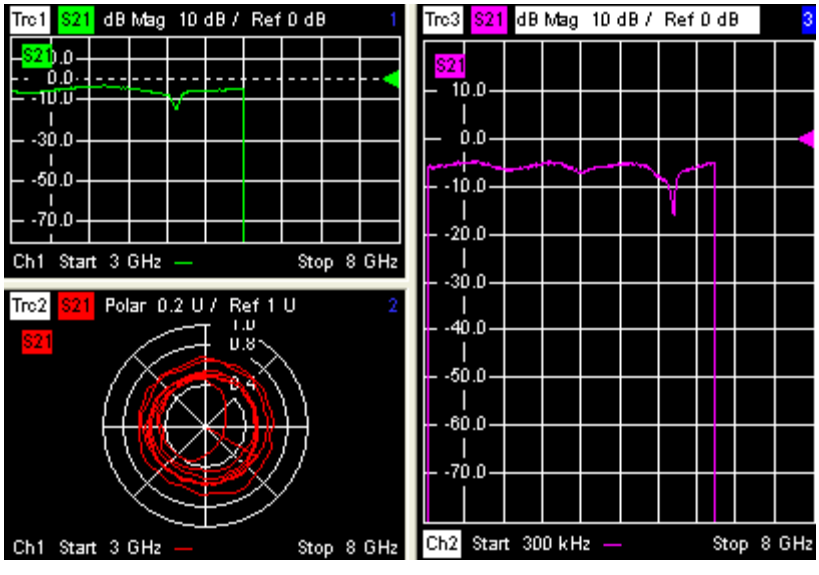
**Stack**



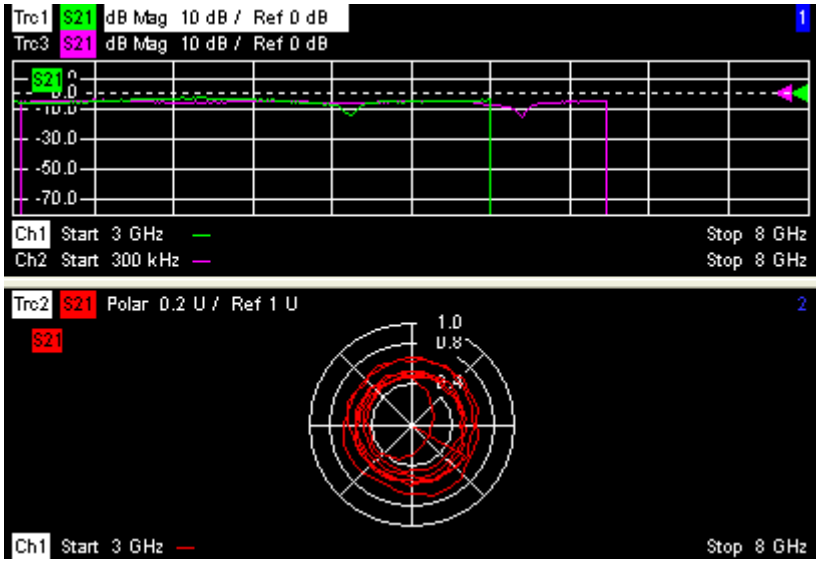

**Tile Horizontally**




**Tile Vertically**



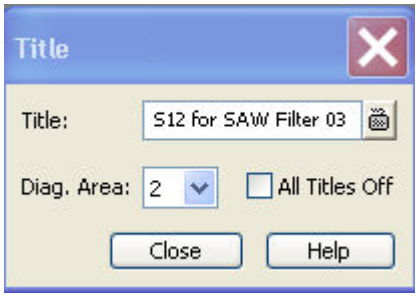
**Rows and Columns (2 | 1):** the 3rd trace is displayed in overlay mode.




**Remote control:** No command, display configuration only.

### Title

Opens a dialog to opens a dialog to define a title and display it in one of the diagram areas.

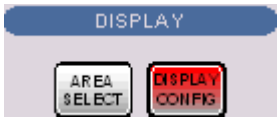


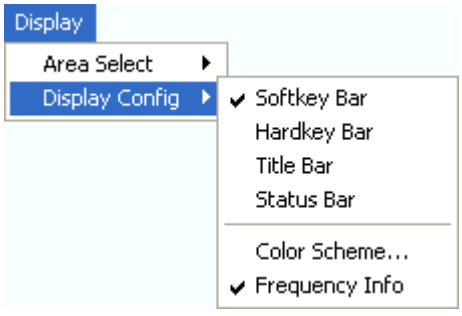

- *Title* provides an input field for the title string. The title may comprise a practically unlimited number of characters and is centered in a line below the top of the diagram area.
- *Diagram Area* provides a drop-down list of all diagram areas of the current setup. The title is assigned to the selected area.
- *All Titles Off* removes the titles from all diagram areas. The titles are hidden but not deleted: Unchecking *All Titles Off* displays the titles again.

**Remote control:**
```
DISPlay:WINDow<Wnd>:TITle:DATA '<title>'
DISPlay:WINDow<Wnd>:TITle:STATe <Boolean>
```

## Display Config

The *Display Config* submenu configures the screen by showing or hiding controls and information elements and controls the appearance of the individual diagrams.




The *Display Config* menu contains the following functions:

- *Softkey Bar* shows or hides the softkey bar at the right edge of the screen.
- *Hardkey Bar* shows or hides the hardkey bar at the left edge of the screen.
- *Title Bar* shows or hides the title bar of the NWA application.
- *Status Bar* shows or hides the status bar across the bottom of the screen.
- *Color Scheme* controls the colors in all diagram areas.
- *Frequency Info* shows or hides the frequency values in the diagram areas.

Hiding the controls and information elements leaves more space for the diagram areas. All elements may be shown or hidden simultaneously. A checkmark next to the menu item indicates that the view element is displayed.

## Softkey Bar

Shows or hides the softkey bar at the right edge of the screen.

The softkey bar shows up to 8 commands of the active menu indicated above softkey no. 1. The figure above shows the upper part of the softkey bar corresponding to the *Trace – Measure* submenu. Pressing the key to the right of a softkey directly activates a submenu, calls up the numeric entry bar or a dialog or initiates an action. See *Softkey Bar* in the introductory chapter for more information.

## Hardkey Bar

Shows or hides the hardkey bar (front panel key bar) at the right edge of the screen (to the left of the softkey bar).

The hardkey bar represents the most commonly used front panel keys of the analyzer. Clicking a key symbol executes the action of the corresponding key.

See *Hardkey Bar* in the introductory chapter for more information.

## Title Bar

Shows or hides the title bar of the NWA application.

The title bar provides the *Control* menu and the buttons to minimize, maximize and close the main application window. In normal operation using the analyzer's display these functions are not needed, so the title bar is hidden by default.

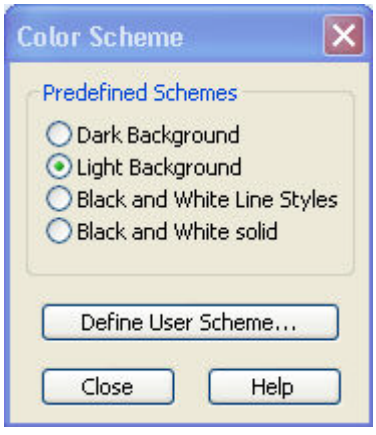
## Status Bar

Shows or hides the status bar across the bottom of the screen.

The status bar describes the current channel (if the setup contains only one channel), the progress of the sweep and the control mode of the analyzer *(LOCAL* or *REMOTE)*.

## Color Scheme

Controls the colors in the diagram areas. Color schemes are global settings and apply to all active setups.




The following *Predefined 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 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.

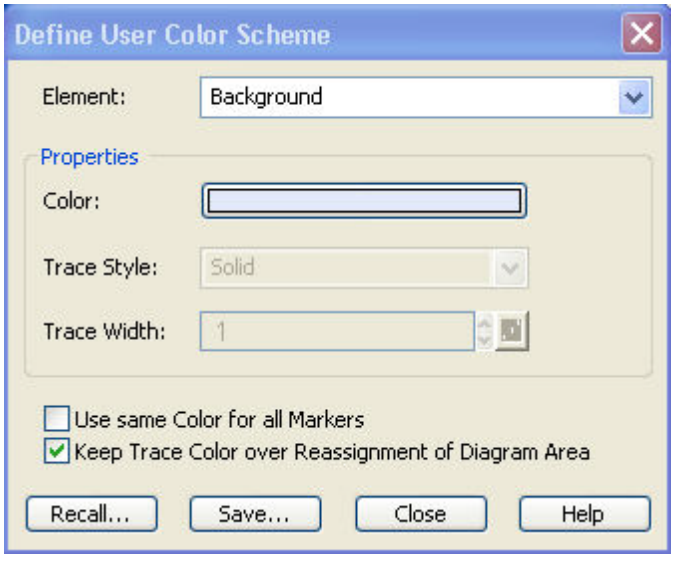
*Define User Scheme...* opens a dialog to modify the predefined schemes, changing the colors and styles of the individual display elements.

Use *Define User Scheme* to create, save and recall your own color scheme.

**Remote control:** `SYSTem:DISPlay:COLor`

## Define User Color Scheme

The *Define User Color Scheme* dialog modifies the predefined color schemes, changing the colors and styles of the individual display elements. User-defined color schemes can be saved to a file for later reuse.




The following control elements change the current color scheme:

- The screen element to be modified is selected from the *Element* drop-down list. The list contains the background and all traces, text elements and lines in the diagrams.
- *Color* opens a standard color dialog to assign a color to the selected element. Use the *What's This?* help button in the dialog to obtain detailed information.
- *Trace Style* and *Trace Width* are enabled if the selected element is a trace.
- *Use same Color for all Markers* allows to select a common marker color, which is independent of the trace colors. To define the common color, select the *Same Color for all Markers* element.
- *Keep Trace Color over Reassignment of Diagram Area* controls the color of traces that are assigned to another diagram area or created together with a new diagram areas; see background information below.

**Effects of *Keep Trace Color...***

The analyzer assigns trace colors according to a predefined scheme, starting with the colors that are easiest to distinguish. On one hand it is advantageous to use the colors at the beginning of the scheme. On the other hand, it is often desirable to use different colors in different diagram areas so that any trace that is moved from one diagram area to another can keep its color.

*Keep Trace Color...* changed between these two alternative color modes as shown below.

| Keep Trace Color | Move trace to another diagram area (Trace Manager) | *Add Trace + Diag. Area (Trace – Trace Select...)* |
|---|---|---|
| ☐ | Trace color changed according to the new diagram area's color scheme | Color scheme of the new diagram area is independent, restarts with the first colors. Consequently the new trace is displayed with a color that has been already used. |
| ☑ | Trace keeps its color | Color scheme of the new diagram area continues color scheme of the previously active area. The new trace is displayed with a new color. |

See also program example for `DISPlay:CMAP<Element>:TRACe:COLor[:STATe]`.

Two buttons at the bottom of the dialog are used to save or recall user-defined color schemes.

- *Save...* opens a *Save As...* dialog to select a color scheme file and save the current settings. Color scheme files are non-editable files with the extension *ColorScheme; the default directory is C:\Rohde&Schwarz\NWA\ColorSchemes.

- *Recall...* opens an *Open File...* dialog to load and apply a color scheme saved before.

To recall a color scheme file (*.ColorScheme) you can also use the Windows Explorer and simply double-click the file or drag and drop the file into the NWA application.

**Remote control:**
```
DISPlay:CMAP<Element>:RGB <red>, <green>, <blue>
[,<trace_style>, <trace_width>]
DISPlay:CMAP<Element>:MARKer[:STATe] ON | OFF
DISPlay:CMAP<Element>:TRACe:COLor[:STATe] ON | OFF
MMEMory:LOAD:CMAP
MMEMory:STORe:CMAP
```

## Frequency Info

Shows or hides all frequency stimulus values in the diagrams. This comprises:

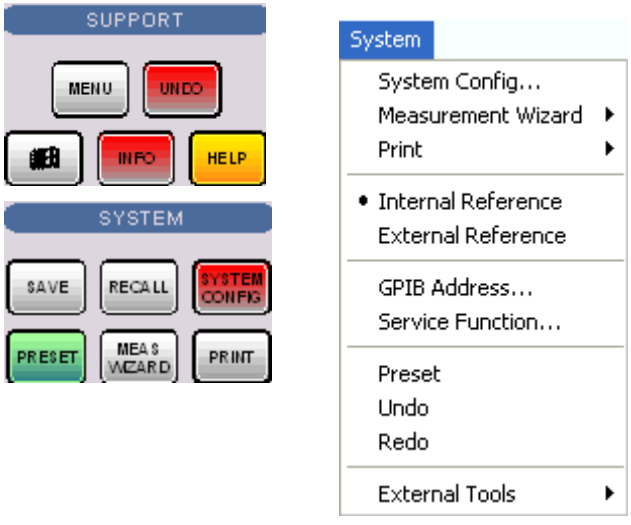
- The frequency stimulus ranges below the diagram area, if a frequency sweep is active.
- The CW frequency in the center below the diagram area, if a power, time or CW mode sweep is active.
- The frequency stimulus values in the marker info field, at the marker position and in the marker table.

The *Frequency Info* setting is valid for frequency and segmented frequency sweeps only.

**Remote control:** `DISPlay:ANNotation:FREQuency[:STATe] ON | OFF`

# System Menu

The *System* menu provides functions to return to a defined instrument state, reverse operations, access service functions and define various system-related settings.

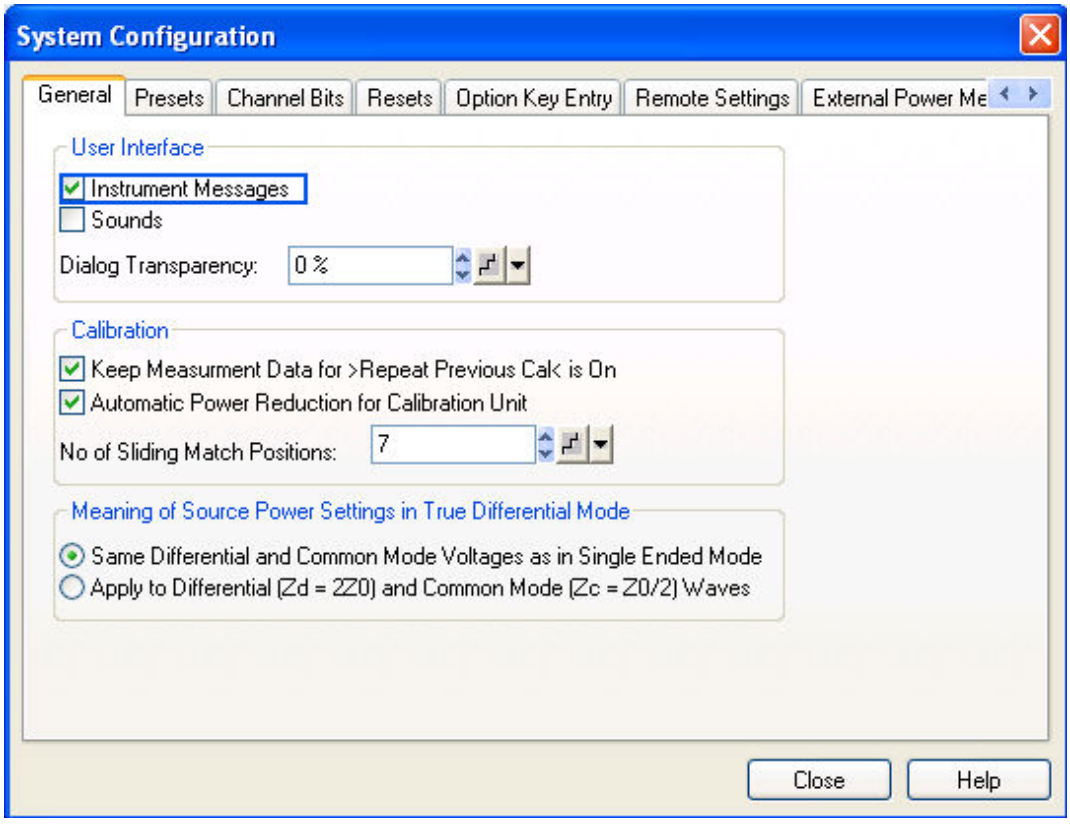



The *System* menu contains the following functions:

- *System Config* opens a dialog to define various system-related settings.
- *Measurement Wizard* calls up a series of dialogs providing a guided standard measurement.
- *Print* opens a submenu to generate a copy of the current screen.
- *Int. Reference* selects the internal reference clock for synchronization.
- *Ext. Reference* selects an external clock signal for synchronization.
- GPIB Address opens the numeric entry bar to define the analyzer's GPIB address.
- *Service Function* opens a dialog to access the service functions of the instrument.
- *Preset* performs a preset of all instrument settings (i.e. all open setups) or of the active setup.
- *Undo* reverses the previous operation.
- *Redo* reverses the action of the *Undo* command.
- *External Tools* opens a submenu with various demo setups and editing tools.

## System Config

Opens a dialog to define various system-related settings.




The *System Config* dialog is divided into the following tabs:

- **General**

  Provides user interface configurations and calibration settings:

  - The two check boxes in the *User Interface* panel switch the instrument messages or acoustic messages on or off. Sounds are generated when the analyzer generates a notice/status message or a warning (alarm sounds) or during calibration. The settings are also valid if the instrument is remote-controlled.
  - *Dialog Transparency* varies the transparency of all dialogs on a scale between 0% and 100%. 0% transparency means that dialogs completely hide the diagram area in the background. With a maximum transparency of 100%, the dialogs are still visible but clearly show the underlying traces and display elements.
  - *Keep Measurement Data for >Repeat Previous Cal<* causes the raw measurement data of the standards to be stored after the calibration is completed. This function is equivalent to the parameter in the *Measure Standards* dialog of the calibration wizard (see detailed description there) but applies to all calibrations.
  - *Automatic Power Reduction for Calibration Unit* sets the source power at all test ports to –10 dBm while an automatic calibration using the calibration unit R&S ZV-Z51 or R&S ZV-Z58 is active. Applying this source power to the ports of the calibration unit ensures best accuracy of the automatic calibration. The source power is reset to its original value (e.g. the preset value of 0 dBm) after the calibration is completed. The automatic power reduction can be deactivated in case that the test setup introduces a large attenuation.
  - *No of Sliding Match Positions* defines the maximum number of different positions to be measured if a sliding match standard (sliding load) is used for calibration. The different positions appear in the *Measure Standards* dialog of the calibration wizard. A sliding match calibration is valid after three calibration sweeps at different match positions; however, calibration sweeps at additional positions can still improve the accuracy. In general 4 to 6 positions are recommended.

- *Meaning of Source Power Settings...* defines the power reference for the true differential mode; see *Source Power in True Differential Mode.*

- **Presets**

  The tab specifies whether a *Preset* affects the *Active Setup* or all open setups *(Instrument)*. A preset does not change any of the properties listed in the *Reset* tab of the *System Config* dialog.

  In the *Preset Configuration* panel, it is possible to specify whether the *Preset* command will perform a factory preset or restore the settings stored to a user preset file. A user preset file is an arbitrary setup (.zvx) file, to be stored using the *File – Save...* command. If the current user preset file is not found (e.g. because it was deleted or moved), the analyzer performs a factory preset.

  *In remote control, a user-defined preset must be initiated using the commands in the SYSTem:PRESet:USER... subsystem. *RST and SYSTem:PRESet always restore the factory preset settings.*

- **Channel Bits**

  Sets a channel-dependent four-bit binary value to control four independent output signals at the USER CONTROL connector (lines 8, 9, 10, 11). 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. Setting the channel bits does not change the analyzer state.

- **Resets**

  Provides several buttons to reset global instrument settings and properties. Global settings (e.g. the data related to global resources) are not affected by an instrument *Preset.*

  The *Use Default Directories* button is inactive in the current firmware version.

- **Option Key Entry**

  Provides an input field for the option key code supplied with each option. A new software option for the network analyzer is enabled by entering the option key and pressing *Install* as described in the dialog.

  Alternatively, it is possible to load the option key from the option key (*.xml) file. Use one of the following methods:

  - Copy the *.xml file to the root directory of a floppy disk or memory stick accessible from your analyzer and press *Install.*
  - Copy the *.xml file to the directory *C*:\Rohde&Schwarz\Nwa of the analyzer's hard disk and press *Install.*

  You can use several *.xml files in order to enable several options simultaneously. *.xml files on the storage medium that are no option key files are ignored.

- **GPIB Settings**

  Specifies the *GPIB Language* and an *ID String* for the analyzer, to be queried via `*IDN?`.

  - If the *DEFAULT* language is activated, the factory ID string *Rohde&Schwarz,ZVB<Max. Freq.-Ports>Port,<Serial_no>,<FW_Version>* (e.g. *Rohde&Schwarz,ZVB8-4Port,1145101010100001,1.70.5)* is set. The bit order for transferred binary data is swapped (`FORMat:BORDer SWAPped`).
  - If the *PNA* is activated, an Agilent-compatible ID string is set. The bit order for transferred binary data is normal.

The ID string can be changed or reset.

*Show Error Messages* activates a information popup box (tooltip) to be displayed whenever the parser encounters an remote control command error. The tooltip appears at the bottom of the remote or manual screen; it is ***not*** displayed for SCPI errors no. –113, *Undefined header:*

The tooltip is to provide information that can be useful for program development and optimization; it does not necessarily indicate that a remote control script is faulty or non-executable.

- External Power Meters
- External Generators
- Frequency Converter
- LXI Configuration

*The System Configuration settings are global and not affected by a preset of the instrument (Preset or `*RST`).*

**Remote control (for *General*)**
```
SYSTem:SOUNd:ALARm[:STATe]
SYSTem:SOUNd:STATus[:STATe]
[SENSe<Ch>:]CORRection:COLLect[:ACQuire]:RSAVe:DEFault
SOURce<Ch>:TDIF:WAVes SENDed | DCMode
```

**Remote control (for preset):**
```
SYSTem:PRESet:SCOPe SINGle | ALL
```

**Remote control (for channel bits):**
```
CONTrol:AUXiliary:C[:DATA]
OUTPut<Ch>:UPORt[:VALue]
```

**Remote control (for GPIB):**
```
FORMat:BORDer NORMal | SWAPped
SYSTem:ERRor:DISPlay
```

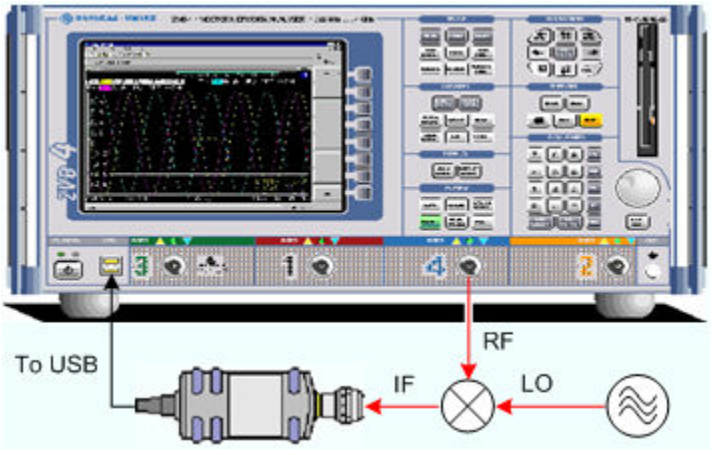
## External Power Meters

Configures external power meters with their connection type and device address. The analyzer supports external R&S NRP-Z sensors as power meters, to be connected via USB interface (the supported types comprise NRP-Z21, NRP-Z11, NRP-Z22, NRP-Z23, NRP-Z24, NRP-Z51, NRP-Z52, NRP-Z55, NRP-Z91).

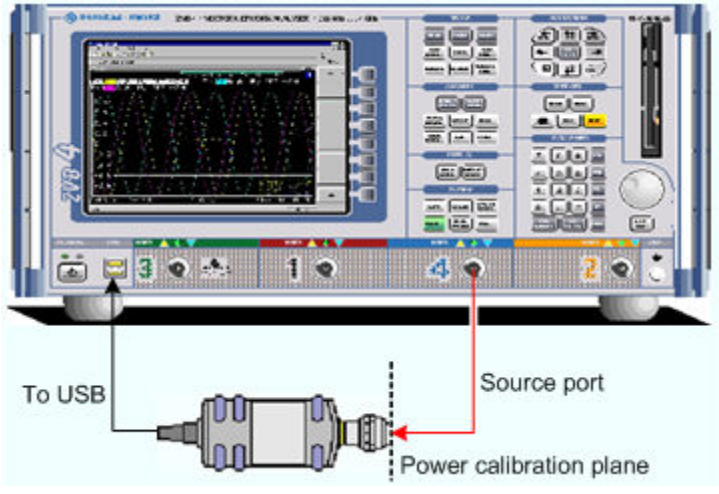
**Use of external power meters, examples**

An external power meter can serve different purposes:

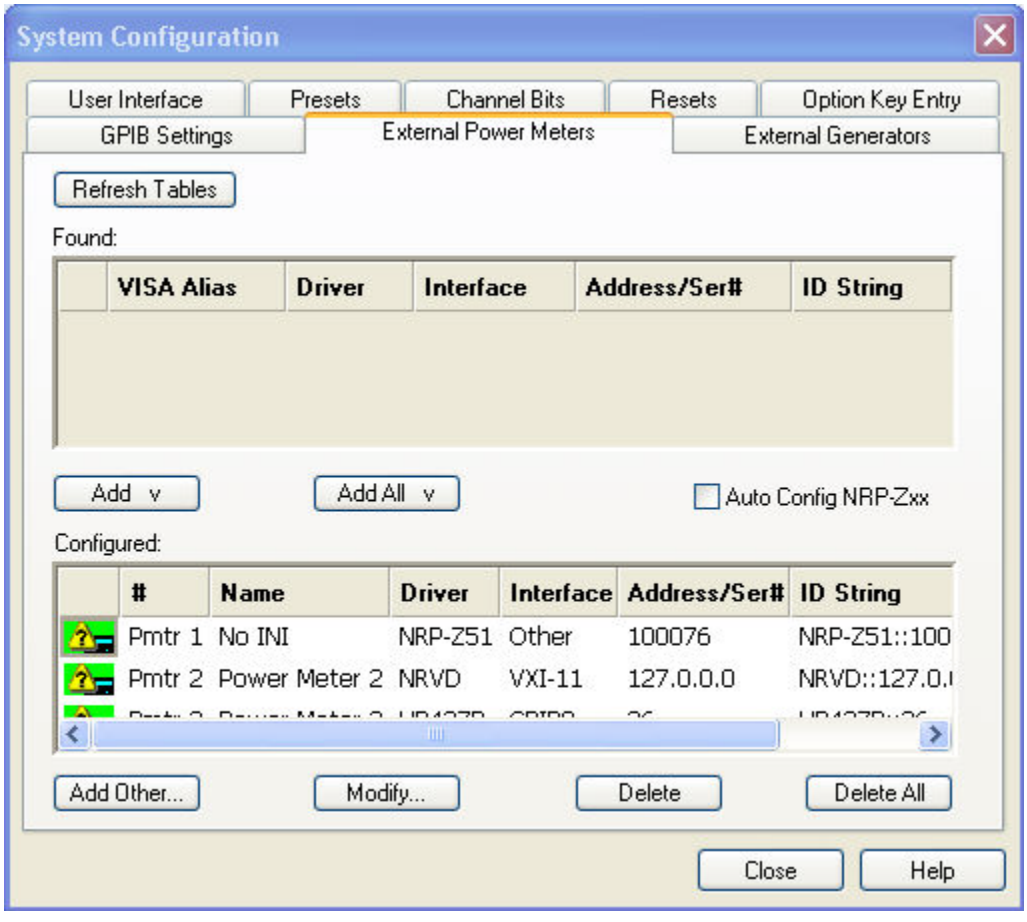
- Extended measurement functionality: Each external power meter represents an additional receive port. External generators increase the number of RF output signals of a DUT that the analyzer can measure simultaneously. They can also provide accurate results for signals at inaccurate or unknown frequencies. 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.




Configured external power meters can be selected as additional receivers in the *Port Configuration* and in the power calibration dialogs.




The *External Power Meters* dialog provides two tables and some additional buttons.

- The upper table *(Found:)* shows all power meters that the analyzer detects to be on line (i.e. connected and switched on). All power meters in the table must have been configured previously, however, they can be deleted in the lower table.
- *Add v* and *Add All v* buttons copy the selected power meter in the upper table to the lower table so that it is possible to modify the power meter settings and select the power meters as receivers for measurements and power calibrations.
- If *Auto Config NRP-Zxx* is selected, the analyzer automatically configures the first power meter detected at one of the USB ports as Pmtr 1. To detect several connected NRP-Zxx power meters and/or add an NRP-Zxx to the lower table, the automatic configuration must be disabled (no risk of arbitrary assignment of power meter numbers).
- The lower table *(Configured:)* shows all configured power meters with their properties. The properties are defined in the *Add/Modify External Power Meter* dialog, to be opened by means of the *Add Other...* or *Modify...* buttons. All configured power meters are available as additional receivers for measurements and power calibrations.

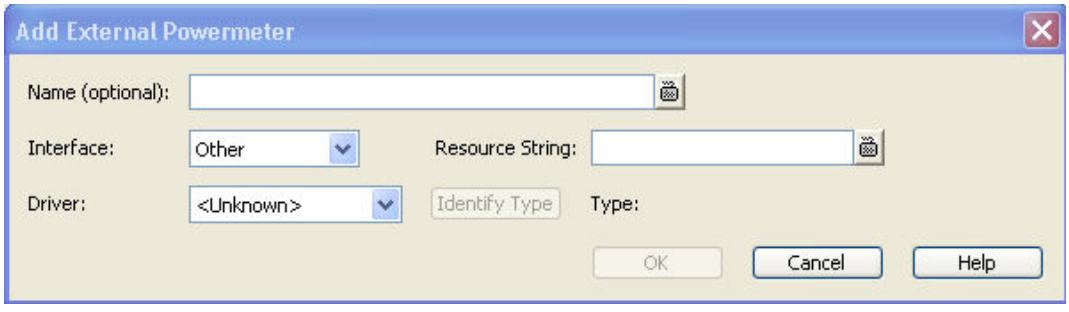
An icon in the first columns of the tables indicates whether the power meters are currently available. The *Found:* table and the icons in the *Configured:* table are automatically updated each time the *External Generators* dialog is opened (to update explicitly use the *Refresh* button). The remaining control elements are self-explanatory.

**Remote control:** `SYSTem:COMMunicate:RDEVice:PMETer<pmet_no>:DEFine`
`SYSTem:COMMunicate:RDEVice:PMETer<pmet_no>:DELete`

## Add/Modify External Power Meter

Selects and configures an external power meter.

*The USB power meters supported at present are configured automatically, all input fields except the optional Name field are disabled. A particular driver and serial number can be selected via remote control.*




**Remote control:** `SYSTem:COMMunicate:RDEVice:GENerator<gen_no>:DEFine`

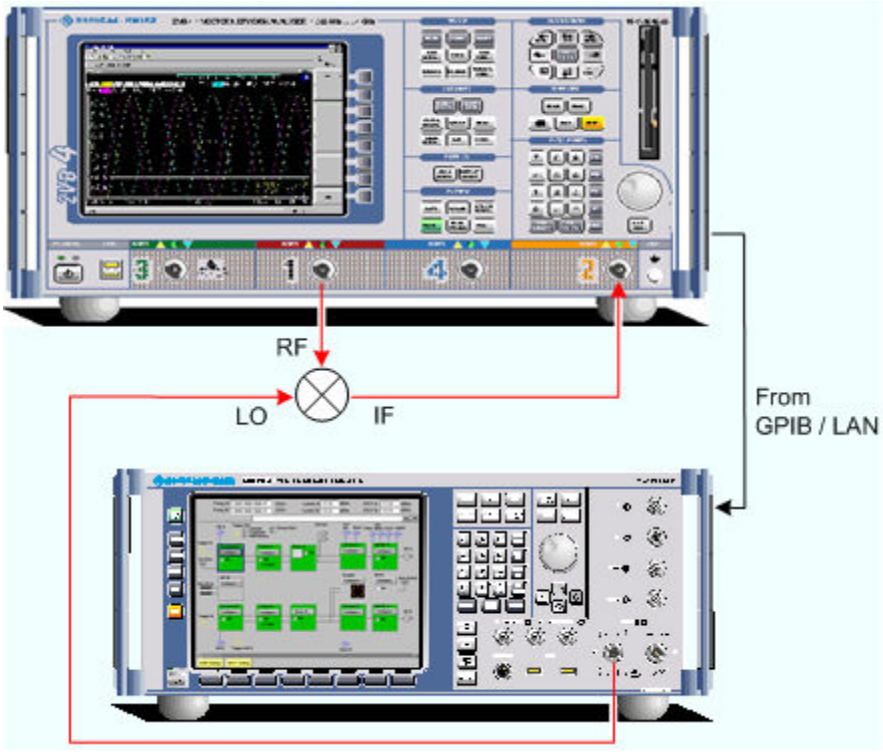
## External Generators

Configures external generators with their connection type and device address.

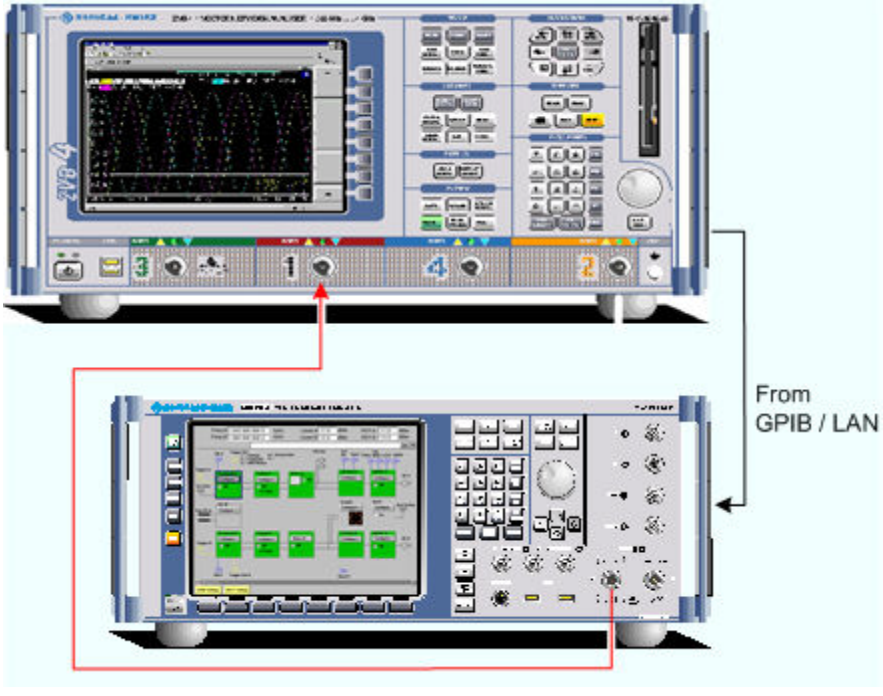
**Use of external generators, examples**

An external generator 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.



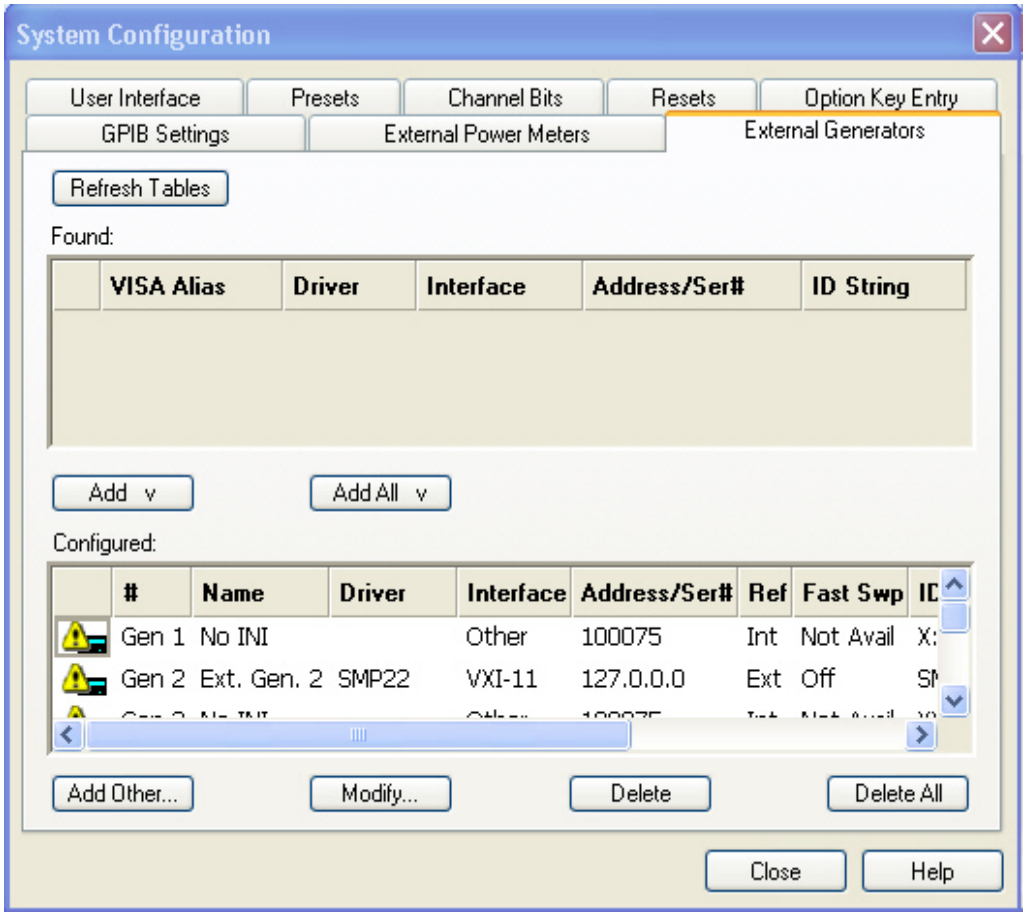

- Power calibration: An external generator can provide the reference signal for a source or receiver calibration. A typical example is a receiver power calibration using a measured wave $b_1$.




Configured external generators can be selected as additional sources in the *More Wave Quantities,More Ratios, Port Configuration,*and in the power calibration dialogs.

*In order to connect external generators via GPIB, VXI-11, or other interface types, it is necessary*

*to install the Virtual Instrument Software Architecture (VISA) library. VISA provides the programming interface between the hardware and the analyzer's application environment.*




The *External Generators* dialog provides two tables and some additional buttons.

- The upper table *(Found:)* shows all generators that the analyzer detects to be on-line (i.e. connected and switched on). All generators in the table must have been configured previously, however, they can be deleted in the lower table.
- *Add v* and *Add All v* buttons copy the selected generator in the upper table to the lower table.
- The lower table *(Configured:)* shows all configured generators with their properties. The properties are defined in the *Add/Modify External Generator* dialog, to be opened by means of the *Add Other...* or *Modify...* buttons. All configured generators are available as additional sources for measurements and power calibrations.

An icon in the first columns of the tables indicates whether the generators are currently available. The *Found:* table and the icons in the *Configured:* table are automatically updated each time the *External Generators* dialog is opened (to update explicitly use the *Refresh* button). The remaining control elements are self-explanatory.

**Icons in the generator tables**

Two different icons describe the status of a generator:

The generator is on-line (connected, switched on, ready to be used)

The generator was detected (upper table) or configured (lower table) but is not on-line

**Remote control:** `SYSTem:COMMunicate:RDEVice:GENerator<gen_no>:DEFine`

SYSTem:COMMunicate:RDEVice:GENerator<gen_no>:DELete

## Add/Modify External Generator

Selects and configures an external generator.

The following control elements select the generator type and its connection:

- *Name* is an optional, arbitrary name associated with the generator. Names appear in the generator lists in the *External Generators* tab and in the selection dialogs for source signals (measurements and power calibration). If no name is specified, the generators are listed as *Gen 1, Gen 2...*
- *Interface* selects an interface type for the connection. In addition to the *GPIB0* and *VXI-11* interface types (for devices connected to the *IEC Bus* or *LAN1/2* connectors on the rear panel of the analyzer, respectively), the analyzer supports any *Other* interface supported by the installed VISA library.
- *GPIB Address* or *Hostname or IP Address* (depending on the *Interface* selection) 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), IP addresses must be compatible with the entries in the VISA library. For an *Other* interface type, the input field contains the *Resource String* from the installed VISA library.
- *Driver* contains a list of all supported generator driver types. The names in the list are identical with the name of the generator driver files (*.gen) stored in the resources\extdev subdirectory of the analyzer's program directory.
- *Identify Type* sends an identification query (IDN?) to the specified device address in order to identify the generator type and select an appropriate driver file. An error message is displayed if the identification fails.

The following control elements provide hardware-related settings:

- *Fast Sweep* enables or disables the fast sweep mode for external generators that support a frequency and level list mode.
- *10 MHz Ref* switches the analyzer to either internal or external frequency reference.

**Fast sweep mode and conditions**

In the 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 power meters.

If *Fast Sweep* is activated, 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 re-transferred 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 CONTROL* 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 CONTROL* 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 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.*

**Reference frequency**

To ensure frequency accuracy and frequency stability in a test setup where different devices represent the signal sources and receivers, it is generally 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* reference frequency mode and operate all other devices in external reference mode using the reference clock signal from the *10 MHz REF* connector on the analyzer's rear panel.
- Use another device as master: Set the analyzer to *External* reference and synchronize it (and all other devices) to the master's reference clock signal, fed in at the *10 MHz REF* connector on the analyzer's rear panel.

The *10 MHz REF* settings in the *Add External Generator* dialog are generator-specific and not overwritten by a change of the global *Internal Reference / External Reference* settings.

**Remote control:** `SYSTem:COMMunicate:RDEVice:GENerator<gen_no>:DEFine`

**Remote control:**
```
SYSTem:COMMunicate:RDEVice:GENerator<gen_no>:DEFine
SYSTem:COMMunicate:RDEVice:GENerator<gen_no>:DELete
```

## Int. Reference

Selects the internal reference clock for synchronization. The analyzer provides a 10 MHz internal reference clock which can be tapped off at the *10 MHz REF* connector at the rear of the instrument in order to synchronize other devices, e.g. signal generators or a second network analyzer of the ZVAB family. See also example for reference frequency settings for external generators.

**Remote control:** `[SENSe<Ch>:]ROSCillator[:SOURce] INTernal`

## Ext. Reference

Selects an external reference clock signal for synchronization. The external 10 MHz reference clock signal must be applied to the *10 MHz REF* connector at the rear of the instrument. The external reference signal must meet the specifications of the data sheet. The internal reference signal is synchronized to the external signal. See also example for reference frequency settings for external generators.

**Remote control:** `[SENSe<Ch>:]ROSCillator[:SOURce] EXTernal`

## GPIB Address

Opens the numeric entry bar to define the analyzer's GPIB address.

The GPIB address must be in the range between 0 and 30.

**Remote control:** `SYSTem:COMMunicate:GPIB[:SELF]:ADDRess`

## Service Function...

Opens a dialog to access the service functions of the instrument. Service functions are password-protected and should be used by a R&S service representative only. Refer to the service manual for more information.

**Remote control (for channel bits):**
`SYSTem:PASSword[:CENable]`
`DIAGnostic:SERVice:FUNCtion`
`DIAGnostic:SERVice:SFUNCtion`

## Preset

Performs a preset of all instrument settings (i.e. all open setups) or of the active setup, depending on the settings in the *System Config* dialog.

**Scope of the preset and exceptions**

A preset may be a factory preset or a user-defined preset and affects either the active setupor all open setups.

It does not change the data related to global resources (cal pool, cal kit data), the position of dialogs, the color scheme of the diagram areas, and the directory and printer settings. All these properties can be reset in the *Reset* tab of the *System Config* dialog.

If you activate *Preset* by mistake, you can use *Undo* in order to restore your previous instrument settings.

**Remote control:**
`*RST; SYSTem:PRESet` (for factory preset)
`SYSTem:PRESet:USER[:STATe] ON | OFF`
`SYSTem:PRESet:USER:NAME '<Setup_file>'` (for user-defined preset)

## Undo

Reverses the last action, if possible. Otherwise, *Undo* is disabled (grayed).

You can use *Undo* even after a *Preset,* in order to restore your own instrument settings.

## Redo

Reverses the action of the *Undo* command. If *Undo* was not used before, *Redo* is disabled (grayed).

## External Tools

Opens a submenu with various demo setups and editing tools:

- *Mouse Keyboard.lnk:* Opens Windows XP's on-screen keyboard.
- *Demo*.vbs:* Shows how to perform the settings for typical measurement tasks.

After running a *vbs file you can modify the demo setup according to your own needs and store it to a *.zvx file for later reuse.

## Converter Control (Option R&S ZVA-K8, for R&S ZVA and R&S ZVT Network Analyzers)

Measurements at frequencies beyond the analyzer's operating range (mm-wave measurements) are achieved by combining a frequency-converting measurement with an external test set (frequency converter). The frequency converter R&S ZVA-Z110 extends the frequency range of R&S ZVA or R&S ZVT network analyzers to a range between 75 GHz and 110 GHz.

The test ports of the frequency converters and the connecting elements are rectangular waveguides. For the R&S ZVA-Z110 converter, the dimensions of the waveguides are according to EIA WR-10, equivalent to RCSR WG-27.

**Required equipment and options**

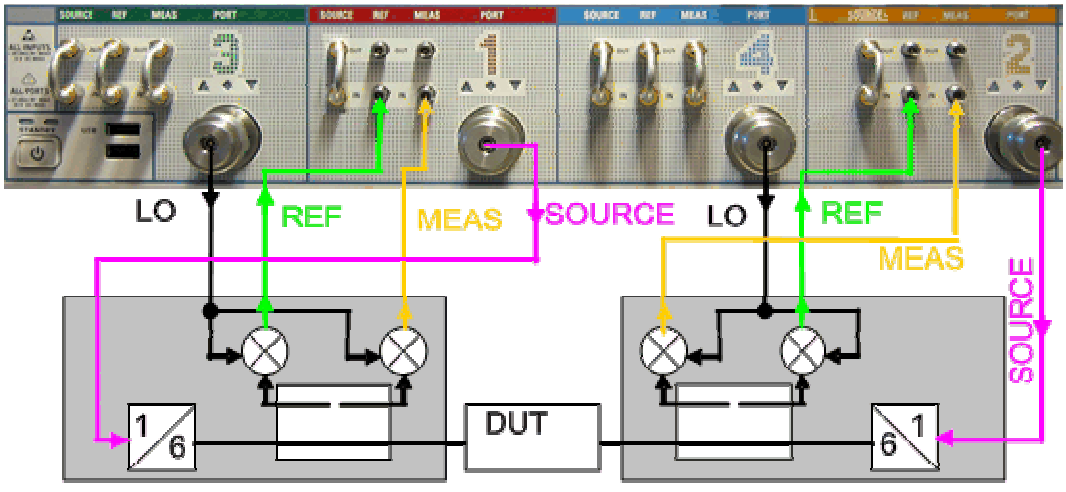
Frequency extension is available on ZVA and ZVT analyzers which are specified for frequencies up to 20 GHz or above (e.g. ZVT 20, ZVA24, ZVA 40). The following additional options and accessories are needed:

- At least one frequency converter R&S ZVA-Z110
- Option R&S ZVA-K8, *Converter Control*. Option R&S ZVA-K8 also comprises option ZVA-K4, *Arbitrary Generator and Receiver Frequencies*.
- Option R&S ZVA<n>-B16, *Direct Generator/Receiver Access* (where <n> denotes the number of test ports of the analyzer)
- A two-port transmission measurement requires two frequency converters plus a four-port network analyzer or a two-port analyzer with an external generator (see External Generators). The external generator must provide frequencies up to 20 GHz or above and sufficient signal power. Reflection measurements can be performed with a two-port analyzer without external generator.
- The external generator must be synchronized to the network analyzer using the 10 MHz reference clock signal. It is not possible to use several external generators simultaneously.
- A suitable calibration kit

### Test Setup and Principle of the Measurement

The frequency converters use frequency multipliers to transform the RF source signal from one of the network analyzer ports into a high-frequency stimulus signal. A dual directional coupler separates the reference and measurement channels from the waveguide test port. A second signal (Local Oscillator, LO) is used for down-conversion of the reference and measurement channels. The LO signal can be provided either by a second analyzer port or by an external generator. The down-converted signals are fed to the REF IN and MEAS IN input connectors of the analyzer port providing the RF source signal.

The schematic test setup for a two-port transmission measurement using four analyzer ports (no external generator) is shown below.

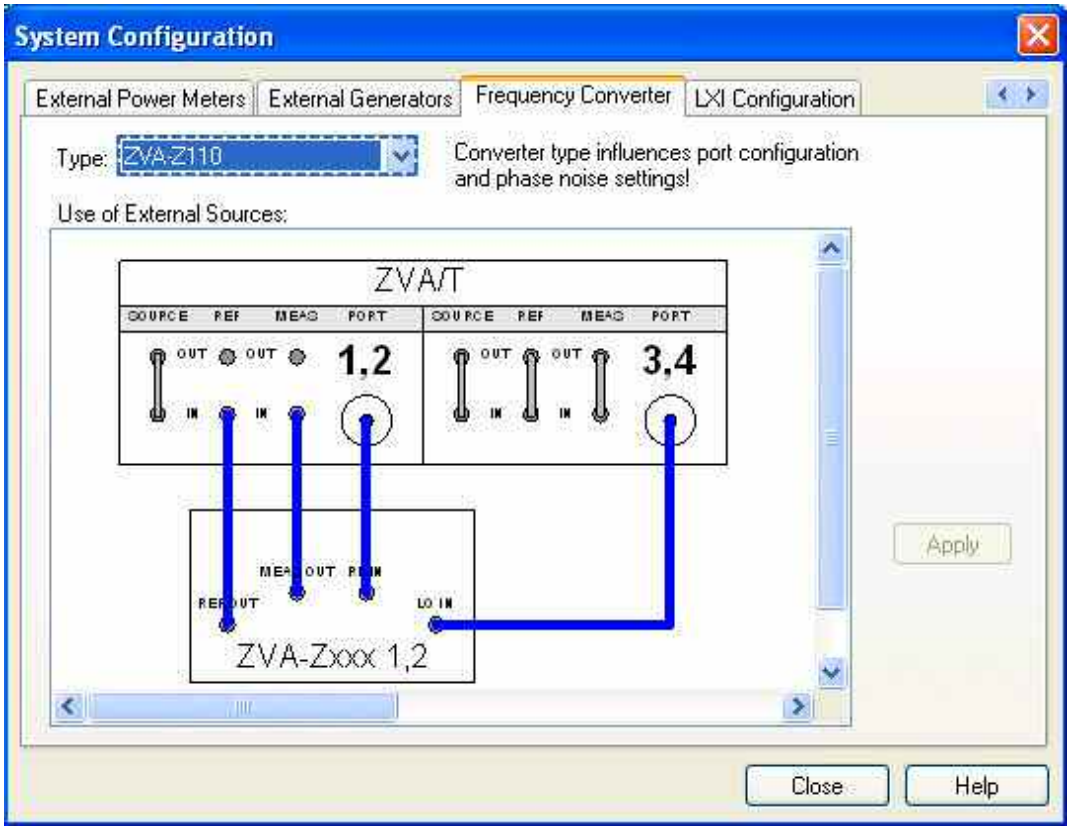



The measurement involves the following steps:

1. Selection of the converter type and of the test setup in the *Frequency Converter* dialog
2. Connection of the frequency converters
3. Calibration using a suitable waveguide kit
4. Connection of the DUT and measurement

## Frequency Converter

The *Frequency Converter* tab belongs to the *System Configuration* menu *(System – System Config).* It selects the frequency converter type and the external test setup.




The dialog provides the following control elements:

- *Type* selects a frequency converter type.
- *Use of External Sources* provides the possible test setups for each converter type. The test setups differ in the way how the LO signal is provided (from the analyzer port or from an external generator).
- *Apply* activates the selected test setup and adjusts the analyzer settings; see information below.

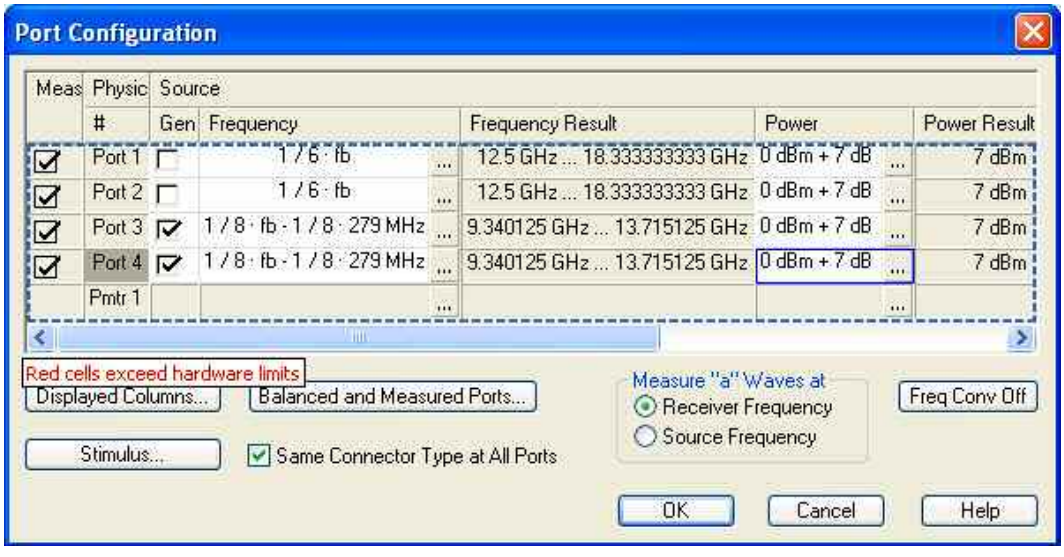
The frequency-converter settings belong to the global resources; they are not changed upon a *Preset* or `*RST`.

**Analyzer settings with active frequency converter**

After a particular converter type and test setup is applied, the analyzer is automatically configured as follows:

- A preset is applied.
- The default frequency range of the selected converter type is set (e.g. 75 GHz to 110 GHz for R&S ZVA-Z110 converters). If no converter is selected, (Type = <NONE>), the default frequency of the network analyzer is set. The *Source* section of the *Port Configuration* table shows the true output frequencies at all analyzer ports and the frequency conversion formulas.
- The output powers at the RF and LO ports are set to optimum values. The output power of the converters can be reduced using the knurled adjusting knob on top of the converter.
- The LO signals are permanently on; the RF source signal is present only during the partial measurements that require the respective test port as a drive port.
- *Low Phase Noise* is enabled.
- *ALC (All Chans)* is disabled.

All signal settings appear in the *Port Configuration* dialog as soon as the frequency converter mode is active. In the example below, a sweep range between 75 GHz and 110 GGHz is selected. The frequency multipliers for the RF signals have a multiplication factor of 6, so the actual output frequency at the RF ports 1 and 2 is between 12.5 GHz and 18.3333333 GHz. The frequency of the LO signal is such that the reference and measurement channels are received at the fixed IF frequency of 279 MHz (shown in the *Receiver* section of the *Port Configuration* dialog). The LO frequency is always below the RF frequency (upper sideband).




**Remote control:** `[SENSe<Ch>:]FREQuency:CONVersion:DEVice:MODE`
`[SENSe<Ch>:]FREQuency:CONVersion:DEVice:NAME`

## Connecting the Frequency Converters

Each frequency converter must be connected to the DUT, the analyzer ports, and the power supply. The DUT is screwed on the waveguide flange at the front of the converter. The remaining connectors are located on the rear panel of the converter:

**Attention!**

**Input powers RF IN and LO IN**

The RF input power at the connectors RF IN and LO IN must not exceed the maximum values quoted in the data sheet. The maximum values are below the maximum RF source power of the network analyzer. The powers in frequency converter mode are in the safe operating range and should normally not be changed.

Always activate the frequency converter mode using the *Frequency Converter* dialog before you connect your converter to the network analyzer.

Connection to the network analyzer comprises the following input and output signals:

1. RF source signal (input): Connect port 1 or port 2 of the analyzer to RF IN of the converters.
2. LO signal (input): Connect port 3 or port 4 of the analyzer to LO IN of the converters. Alternative: Connect an external generator signal to LO IN; see *Frequency Converter* dialog.
3. Reference signal (output): Connect REF OUT of the converters to the REF IN connector of the analyzer port providing the RF source signal. The reference signal corresponds to the incident wave (a-wave) applied to the input of the DUT.
4. Measurement signal (output): Connect MEAS OUT of the converters to the MEAS IN connector of the analyzer port providing the RF source signal. The measured signal corresponds to the b-wave caused due to reflection or transmission at the DUT.

*The input signal at the frequency converters must be relatively stable over the entire sweep range. Use high-quality cables for the connection of the RF and LO input signals. Appropriate cables for the REF OUT and MEAS OUT signals are supplied with the frequency converters.*

*In view of the port arrangement at the front panel of the analyzer, it is advantageous to use test ports 1 and 2 as RF sources, ports 3 and 4 as LO sources. The frequencies of the RF and LO signal must be independent, therefore do not combine a pair of ports as RF and LO sources that is supplied by a common generator (ports 1 and 2, 3 and 4; see Coupled test ports).*

To supply the frequency converter, connect the external DC power supply provided with the converter to the *9 V / 0.5 A* DC input. The power supply supports input AC voltages between 100 V and 240 V and frequencies between 47 Hz and 63 Hz. A complete test setup for a 2-port transmission measurement is shown below.

**Attention!** **Supply voltage and power**

The input voltage and current must not exceed the maximum values according to the rear panel labeling or the data sheet.

Always use the DC power supply included in delivery to power your frequency converter.

## Calibration

Due to the physical properties of the mm-waves and the waveguides, measurements with frequency converters require a special calibration kit. Rohde & Schwarz offers kits for this purpose, e.g. the calibration kit R&S ZV-WR10.

**Standards for waveguide calibration, calibration data**

The calibration kit R&S ZV-WR10 contains the following standards:

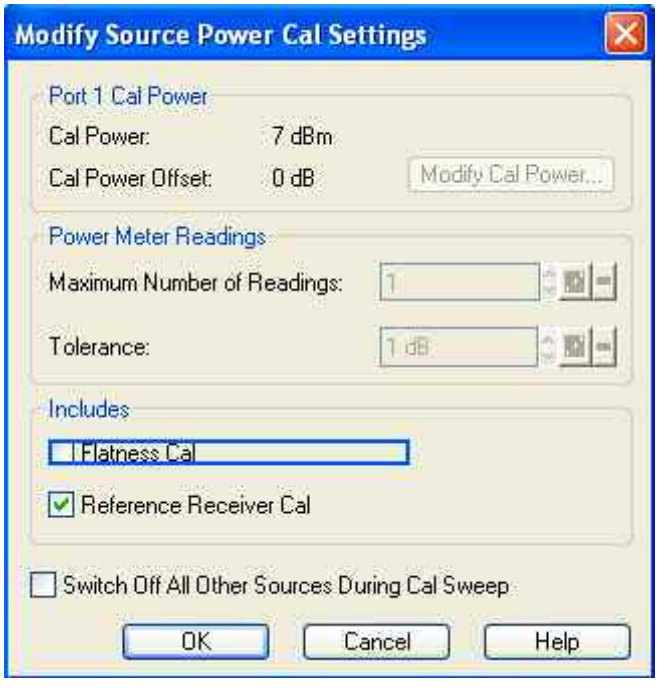
- A Short standard which also serves as a Reflect standard.
- A Match standard.
- A Shim that contains a short transmission line causing a $\lambda/4$ phase shift at the center frequency of the converter's operating range. At this center frequency, the Shim causes a 180 deg phase shift of the reflection coefficient.
- The calibration data of the standards are sufficiently reproducible over the entire operating range of the frequency converters. They are stored in the network analyzer and do not have to be loaded separately for each individual calibration kit.
- In addition to the standards listed above, Sliding Match standards for the relevant frequency range are available. The following table gives an overview of the available standards for rectangular waveguide calibration.

| Standard Type | Technical implementation |
|---|---|
| Open | Not available in waveguide technology, replaced by offset short |
| Short | Contained in the calibration kit |
| Offset short | Short + Shim, both contained in the calibration kit |
| Match | Contained in the calibration kit |
| Sliding match | Available |
| Reflect | Short |
| Through | Direct through connection of the two waveguide flanges |
| Line1 | Through connection of the two waveguide flanges with Shim in-between.<br>In view of the bandwidth of the frequency converters,<br>line 2 is not needed. |
| Attenuation | Not in the calibration kit |
| Symm. network | Not in the calibration kit |

The standards in the calibration kit allow all one-port and two-port calibration types supported by the network analyzer except TNA.

The output power of the frequency converter can be set manually (at the converter) only, therefore the standard source power calibration eliminating frequency response errors in the signal path between the source and the reference plane (external power meter) is not possible. A power calibration of the reference receiver (a-wave) using an external power meter, however, is possible and recommended for measurements concerning the wave quantities a and b. Proceed as follows:

1. Ensure that the output power of the frequency converter is not attenuated (adjust the knurled knob at the top of the converter to 2 mm).
2. Connect an appropriate external waveguide power meter to the waveguide flange and open the *Channel – Calibration – Start Power Cal – Source Power Cal* dialog (see *Source Power Cal)*.
3. Click Modify Settings and disable *Flatness Cal*, leaving *Reference Receiver Cal* checked.




4. Start the calibration sweep.

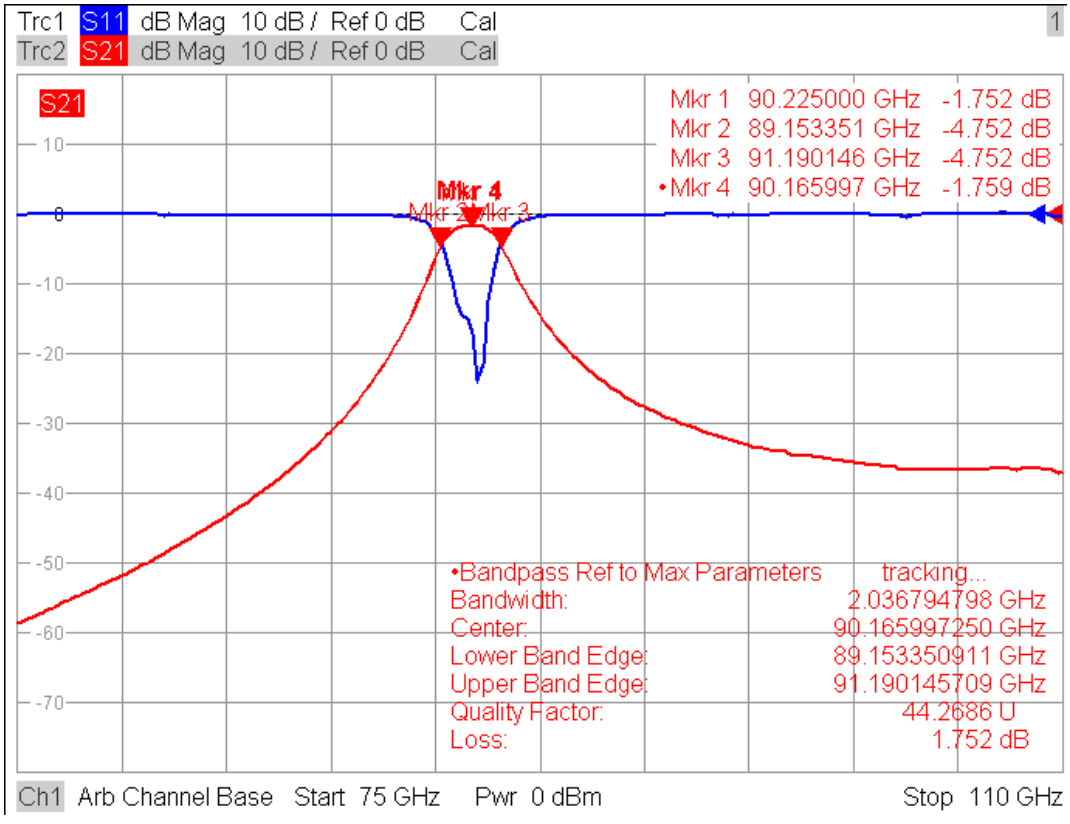
This power calibration procedure ensures a reasonable accuracy of the reference power readings over a wide range of converter output powers (i.e. even if the adjusting knob is used to reduce the powers).

A *Receiver Power Calibration* of the b-waves (without external power meter) is always possible.

## Measurement

After system error correction and – possibly – power calibration, the mm-wave measurement can be performed like any other network analyzer measurement. The *Stimulus* settings determine the sweep range of the converted signals (i.e the input and output frequencies at the DUT ports). All measured quantities (S-parameters, wave quantities, ratios etc.) and other trace settings are available. The following restrictions hold for measurements with external frequency converters:

- The measurement is performed at fixed RF source and LO power. No power sweep is possible.
- To reduce the actual output power of the converters (e.g. for measuring wave quantities or testing compression effects), turn the adjusting knob on top of the converters clockwise. You will see an effect on the output power within the last 2 mm of the adjustable range.
- Note the special analyzer settings listed in section *Frequency Converter.*




## LXI Configuration

LAN eXtensions for Instrumentation (LXI) is an instrumentation platform for measuring instruments and test systems that is based on standard Ethernet technology. LXI is intended to be the LAN-based successor to GPIB, combining the advantages of Ethernet with the simplicity and familiarity of GPIB.

The network analyzer complies with LXI class C. It provides the following LXI-related functionality:

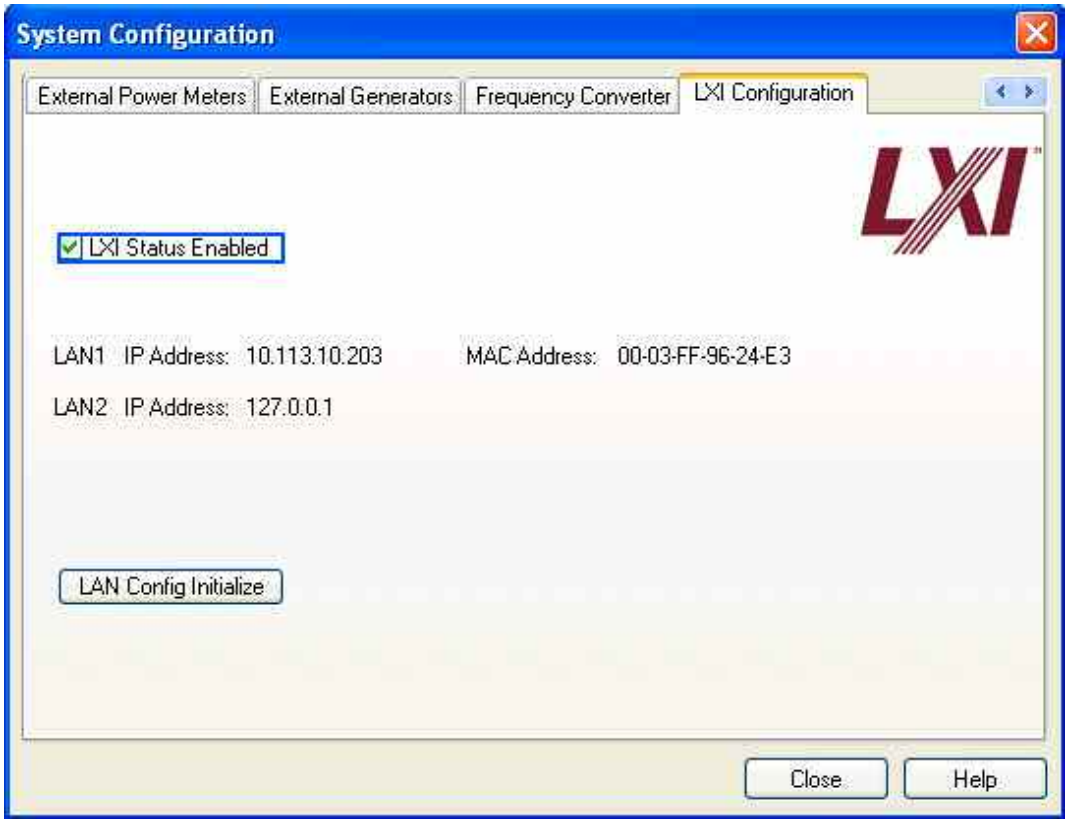
- Integrated *LXI Configuration* dialog for LXI activation and automatic reset of the LAN configuration (LCI)
- Standardized LAN and web browser interface providing information about the instruments and the web settings.
- LAN configuration from the browser interface.
- Ping for connection test.

*The LXI functionality requires a Windows XP operating system that has been upgraded to service pack 2. Please contact your Rohde & Schwarz service representative if your network analyzer's operating system needs an upgrade. To check the version of your operating system, open the System – General dialog in the Control Panel.*

For information about the LXI standard refer to the LXI website at http://www.lxistandard.org/home/. See also "News from Rohde & Schwarz, article 2006/II - 190".

## LXI Configuration

Provides basic LXI functions for the analyzer. *LXI Configuration* is a tab of the *System – System Configuration* dialog.




- *LXI Status Enabled* switches the LXI logo in the status bar on or off.
- *LAN Config Initialize* initiates the hardware network configuration reset mechanism (LAN Configuration Initialize, LCI) for the network analyzer.

**Default state of the network settings**

An LCI places the following parameters to a default state.

| Parameter | Value |
|---|---|
| TCP/IP Mode | DHCP + Auto IP Address |
| ICMP Ping | Enabled |
| Dynamic DNS | Enabled |
| Password for LAN configuration | LxiWebIfc |

The LAN settings are configured using the analyzer's LXI Browser Interface.

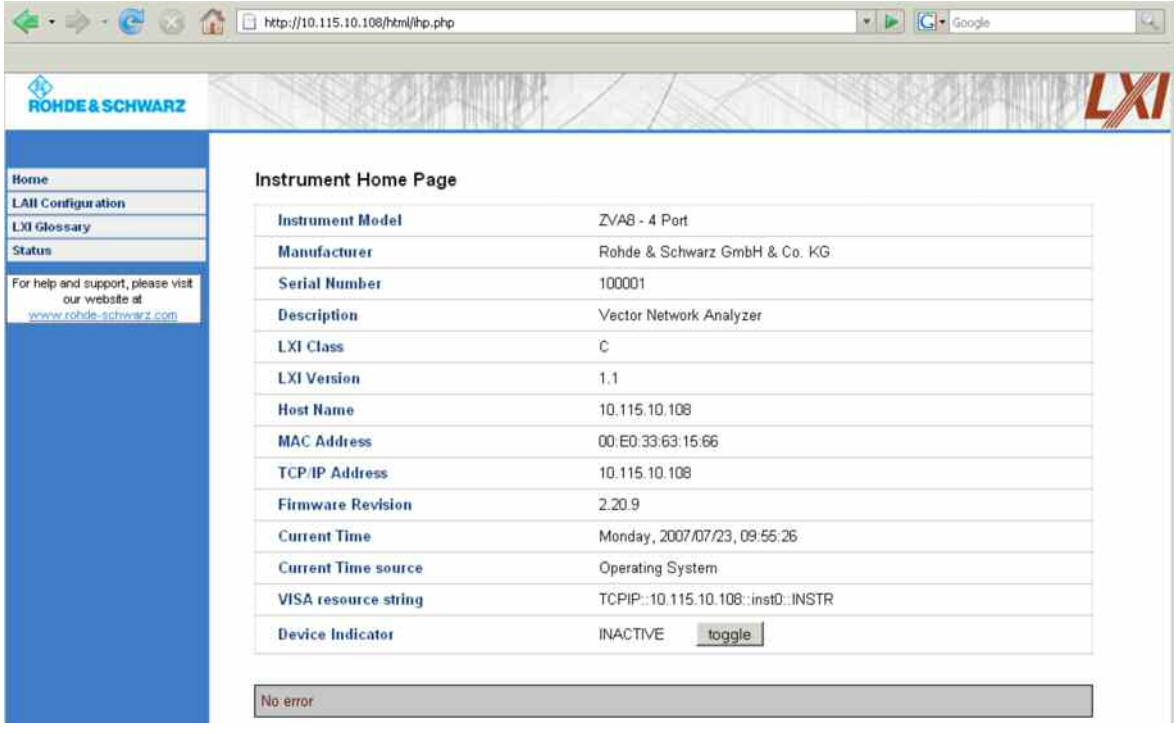
**Remote control:** –

## LXI Browser Interface

The analyzer's LXI browser interface works correctly with all W3C compliant browsers. Typing the instrument's host name or IP address in the address field of the browser, e.g.

*http://10.113.10.203*

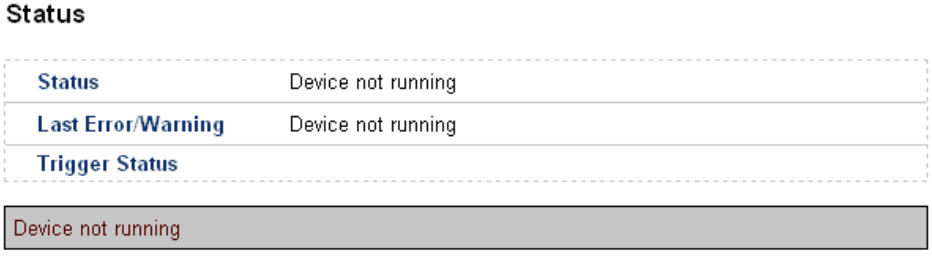
opens the welcome page with various information about the network analyzer.




The welcome page contains the following control and information elements:

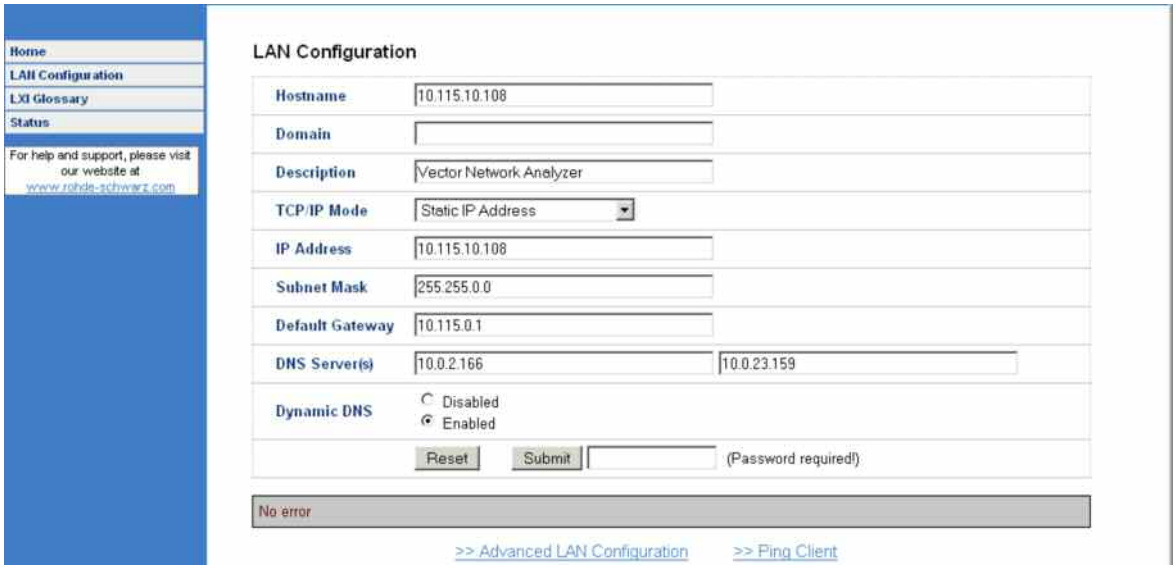
- The *LAN Configuration* button in the navigation pane on the left side opens the LAN Configuration page. The buttons below are for future extensions.
- The *Instrument Home Page* displays the device information required by the LXI standard including the VISA resource string in read-only format. Below, the *Device Indicator* causes the LXI logo in the status bar of the analyzer to blink (if active). The *Device Indicator* setting is not password-protected.

While the network analyzer is off-line (e.g. switched off) the browser opens the following simplified welcome page:



## LAN Configuration

The *LAN Configuration* page of the analyzer's browser interface provides all mandatory LAN parameters.

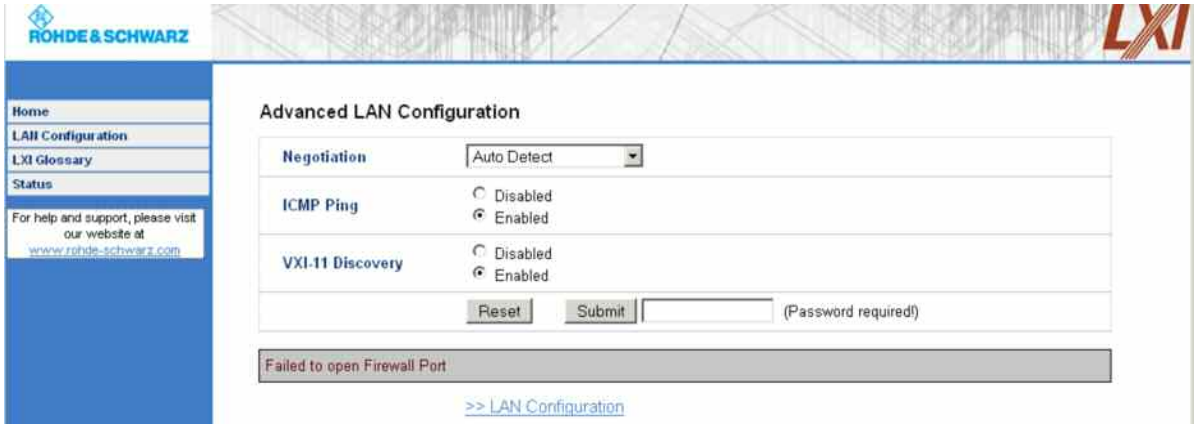



The *TCP/IP Mode* configuration field controls how the IP address for the instrument gets assigned (see also *Assigning an IP Address)*. For the manual configuration mode, the static IP address, subnet mask, and default gateway are used to configure the LAN. The automatic configuration mode uses DHCP server or Dynamic Link Local Addressing (Automatic IP) to obtain the instrument IP address.

*Changing the LAN configuration is password-protected. The password reads **LxiWebIfc** (notice upper and lower case characters). This password cannot be changed in the current firmware version.*

The links across the bottom of the *LAN Configuration* page open other pages:

- *Ping Client* provides the ping utility to verify the connection.
- *Advanced LAN Configuration* provides LAN settings that are not declared mandatory by the LXI standard; see below .




The *Advanced LAN Configuration* parameters are used as follows:

- The *Negotiation* configuration field provides different Ethernet speed and duplex mode settings. In general, the *Auto Detect* mode is sufficient.

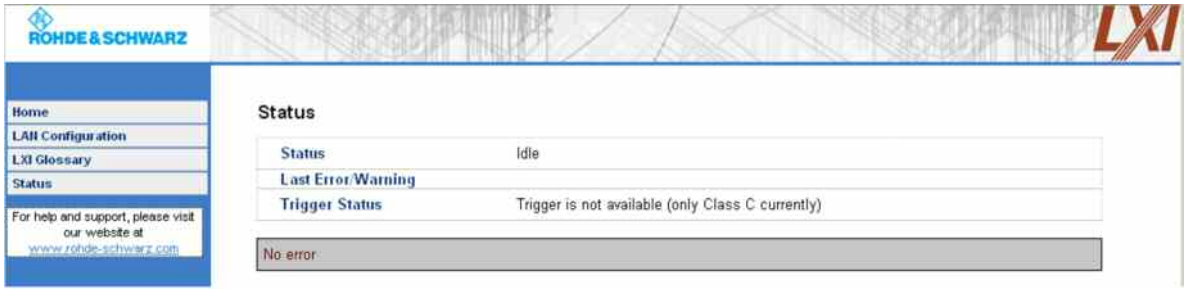
- *ICMP Ping* must be enabled to use the ping utility.
- *VXI-11* is the protocol that is used for discovery of the analyzer in the LAN. According to the standard, LXI devices must use VXI-11 to provide a discovery mechanism; other additional discovery mechanisms are permitted.

## LXI Glossary

*LXI Glossary* opens a document with a glossary of terms related to the LXI standard.

## Status

The *Status* page of the analyzer's browser interface displays status information.

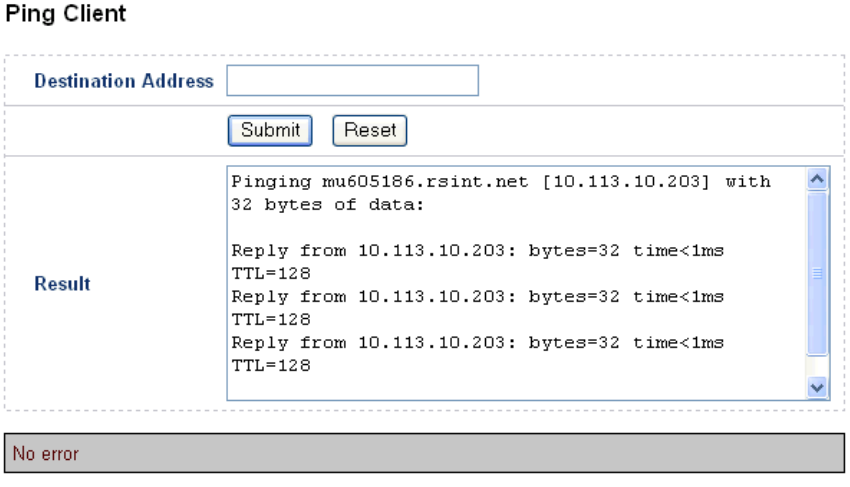



## Ping

Ping is a utility that verifies connection between the controller and the analyzer. The ping command uses the ICMP echo request and echo reply packets to determine whether the LAN connection is functional. Ping is useful for diagnosing IP network or router failures.

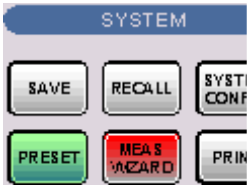
The ping utility is not password-protected. To initiate a ping,

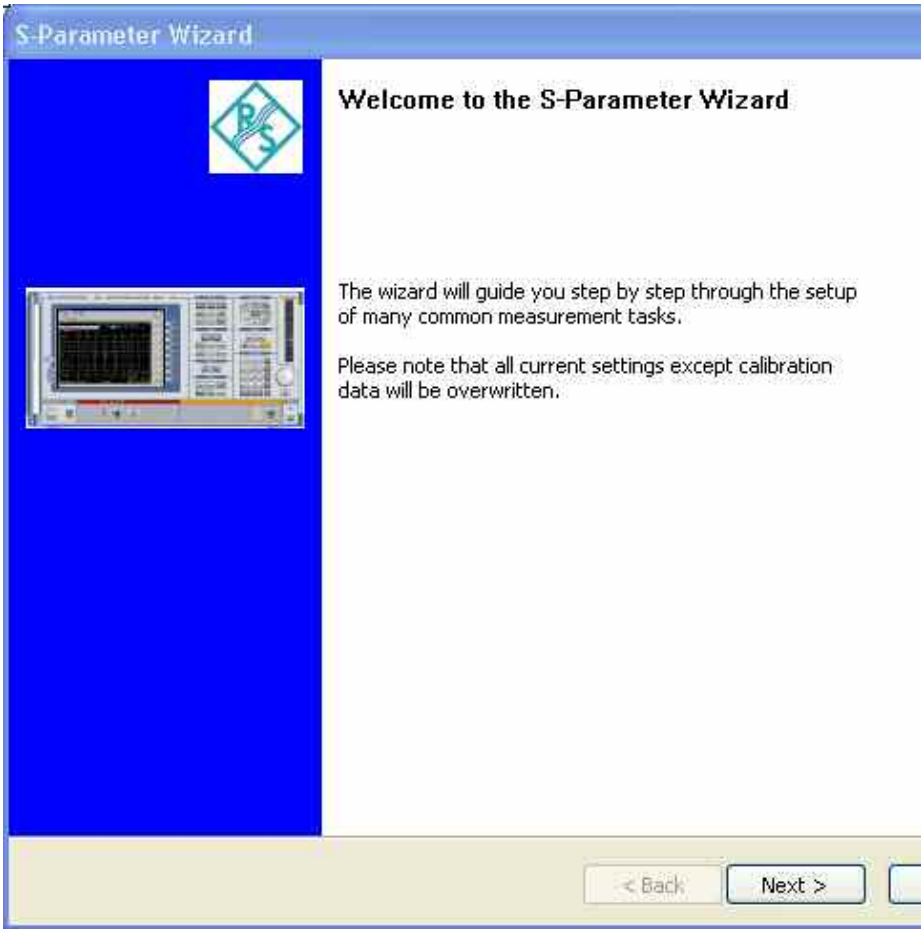
1. Enable *ICMP Ping* on the *Advanced LAN Configuration* page.
2. Enter the analyzer's IP address **without the ping command and without any further parameters** into the *Destination Address* field (e.g. *10.113.10.203*).
3. Press *Submit.*



# Measurement Wizard

The *Measurement Wizard* menu consists of a series of dialogs providing the settings for a standard S-parameter measurement in a frequency sweep.




The measurement comprises the following stages:

5. **Select the test setup**

   Choose the single-ended or balanced ports of the analyzer according to the port configuration of your DUT and connect the DUT to the selected analyzer ports. This dialog corresponds to the *Predefined Configs* tab of the *Balanced and Measured Ports* dialog.

   TIP The default test setup corresponds to a standard 2-port S-parameter measurement. You do not need to select another test setup unless you want to measure multiport parameters or apply a unbalance-balance conversion in order to obtain mixed mode parameters.

6. **Define port impedances**

   Assign reference impedances to all physical and balanced test ports selected in the previous step. The reference impedances can be complex.

   The values for balanced ports are by pressing the ... buttons in a table of the following shape:

| Physical Port | Ref Impedance |
|---|---|
| 1 | 50 Ω + j0 Ω ... |
| 2 | 50 Ω + j0 Ω ... |
| 3 | 50 Ω + j0 Ω ... |
| 4 | 50 Ω + j0 Ω ... |

The buttons open the *Reference Impedance of Physical Port* dialog.

The values for balanced ports are by pressing the ... buttons in a table of the following shape:

| Physical Port | Logical Port | Ref Impedance |
|---|---|---|
| 1 | 1 | DM: 100 Ω + j0 Ω<br>CM: 25 Ω + j0 Ω ... |
| 3 | | |
| 2 | 2 | DM: 100 Ω + j0 Ω<br>CM: 25 Ω + j0 Ω ... |
| 4 | | |

The buttons open the *Reference Impedance of Logical Port* dialog.

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 these value unless you want to renormalize the port impedances; see *Reference Impedances*.

7. **Select the measurement parameters and the diagram areas**

   Choose an S-parameter or a group of S-parameters to be measured and displayed. The measurement parameters are displayed in separate diagram areas: Cartesian diagram areas are used for transmission S-parameters, Cartesian or Smith diagrams for reflection parameters.

   The selections to be made depend on the test setup selected in the previous stage.

   In case of an error in a previous stage, you can always use the *Back* button and correct your settings.

8. **Select the sweep settings**

   Choose the frequency range and the number of measurement points per sweep. The sweep range is defined by two values (start and stop frequency or center frequency and span). The measurement points are equidistant across the sweep range. Increasing the number of points also increases the measurement time per single sweep.

9. **Select the measurement bandwidth and source power**

   Choose a typical measurement bandwidth and one of two typical source power values. A smaller measurement bandwidth increases the dynamic range but slows down the measurement. A smaller source power protects the input port of the analyzer from being overdriven if an active DUT with high gain is measured.

   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 a $S_{21}$ trace at minimum (–40 dBm) 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 power for an active DUT ensures no compression if an active DUT with 20 dB gain is measured. If the actual gain of the DUT is more than 30 dB, then the default source power of –20 dB is still too high and needs to be changed after finishing the wizard.

10. **Perform a calibration (optional)**

    Call up the calibration wizard and perform a calibration. The calibration type is a full n-port calibration, where n is the number of analyzer ports selected in the first stage of the calibration wizard. This calibration can be performed as well using automatic calibration (if accessory R&S ZV-Z51/-Z52 or R&S ZV-Z58, Calibration Unit, is available).

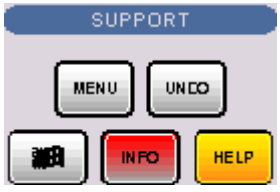
You can skip the automatic calibration stage (select *Finish now without calibration)* if

- A valid calibration is already assigned to the active channel
- You want to apply a valid calibration stored in the cal pool
- You want to perform the calibration manually, e.g. because you wish to select another calibration type.
- You don't want to use a calibration, e.g. because the factory calibration is accurate enough for your measurement.

*In order to obtain a predictable result the measurement wizard has to reset all settings except the current calibration data. Store your setup if you don't want to lose the current configuration.*

# Info Menu

The *Info* menu gives access to information about the instrument and its operation. All functions are primarily intended for error diagnostic and service purposes. Many *Info* dialogs also display softkeys to *Print* the contents, save them to a file *(Print to File)* or *Close* the dialog.



The *Info* menu contains the following functions:

- *Setup Info* displays the channel and trace settings of the active setup and the main characteristics of the instrument including its IP address.
- *Option Info* shows the installed software options with their type and name, the option key and key type, the duration of activation, and the expiry date (if applicable).
- *Hardware Info* gives an overview of the analyzer's hardware configuration.
- *Selftest...* displays a list of the automatic selftests of the analyzer together with the current results.
- *Error Log...* opens a chronological record of errors that occurred in the current and in previous sessions. While the error log is open, additional softkeys to print, close or clear (delete) the log are provided. The deleted error log shows the message *No errors found.*

# Window Menu

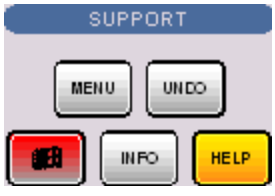
The *Window* menu provides standard Windows™ functions to arrange different windows on the screen.

**Windows**

A window is a rectangular portion of the screen showing all diagram areas of a particular setup. Windows are limited by a blue frame with several icons. The analyzer uses standard windows provided by the operating system.

- A window can show a practically unlimited number of diagram areas.
- The icons on the frame of the window are used to minimize, maximize or close windows.
- The functions of the *Windows* menu are used to arrange windows automatically and select a particular window.
- New windows with new setups are created using the New command.

The contents of the windows are explained in section Display Menu.



The *Window* menu contains the following functions:

- *New* creates a new setup and opens a new setup window.
- *Close* closes an opened setup window.
- *Cascade* arranges all opened windows in an overlapped fashion.
- *Tile* arranges all opened windows in non-overlapped tiles.
- *Maximize* maximizes the active window to occupy the whole screen.
- *1 Set<n>.zvx...* activates a specific window with a specific setup

## New

Creates a new setup and opens a new setup window. The new setup is named *Setup<n>* where <n> is the current number for all created setups.

This command is equivalent to File – New. To open an existing setup, select File – Open / Recall. To rename a setup, use *File – Save As*.

**Remote control:** `MEMory:DEFine "<setup_name>"`

## Close

Closes an opened setup window. The analyzer suggests to save changes to the setup before closing it. If a setup is closed without saving, all changes made since the last time it was saved are lost.

This command is equivalent to File – Close.

**Remote control:** `MEMory:DELete[:NAME] "<setup_name>"`

## Cascade

Arranges all opened windows in an overlapped fashion, showing each one's title bar. The active window is placed in the foreground.

**Remote control:** None (screen configuration only)

## Tile

Arranges all opened windows in non-overlapped tiles.

**Remote control:** None (screen configuration only)

## Maximize

Maximizes the active setup window to occupy the whole screen.

**Remote control:** None (screen configuration only)

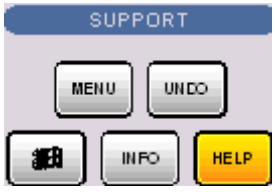
## 1 Set<n>.zvx

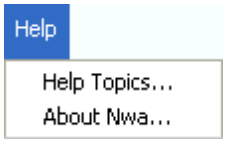
Displays a list of currently open setup windows. A checkmark appears in front of the setup name of the active window. Selecting a setup from this list makes its window active and places it in the foreground.

**Remote control:** `MEMory:CAT?`

# Help Menu

The *Help* menu provides assistance with the network analyzer and its operation.




The *Help* menu contains the following functions:

- *Help Topics* opens this help system.
- *About Nwa...* opens a dialog to retrieve information about the network analyzer and the current firmware version.

## Help Topics

Opens this help system. The help file is opened with the *Welcome* topic and remembers its last size, position and default navigation tab *(Contents, Index,...).* For more information see *About this Help*.

## About Nwa...

Opens a dialog to retrieve information about the network analyzer and the current firmware version. *OK* closes the dialog.

# Contents of Chapter 5


**5 Remote Control ..... 5.1**

**Special Terms and Notation ..... 5.1**

**Remote Control: Introduction ..... 5.3**

GPIB Explorer ..... 5.3

Switchover to Remote Control ..... 5.5

Setting the Device Address ..... 5.6

Combining Manual and Remote Control ..... 5.7

Return to Manual Operation ..... 5.8

Messages ..... 5.8

GPIB Bus Interface Messages ..... 5.9
RSIB Interface Messages ..... 5.9
Device Messages (Commands and Device Responses) ..... 5.9

SCPI Command Structure and Syntax ..... 5.9

Common Commands ..... 5.10
Device-Specific Commands ..... 5.10
Structure of a Command Line ..... 5.11

Responses to Queries ..... 5.12

SCPI Parameters ..... 5.13

Numeric Values ..... 5.13
Boolean Parameters ..... 5.13
Text Parameters ..... 5.14
Strings 5.14
Block Data Format ..... 5.14
Overview of Syntax Elements ..... 5.14

**Basic Remote Control Concepts ..... 5.16**

Traces, Channels, and Diagram Areas ..... 5.16

Active Traces in Remote Control ..... 5.17

Initiating Measurements, Speed Considerations ..... 5.18

**Command Processing ..... 5.19**

Input Unit ..... 5.19
Command Recognition ..... 5.19
Data Set and Instrument Hardware ..... 5.20
Status Reporting System ..... 5.20
Output Unit ..... 5.20
Command Sequence and Command Synchronization ..... 5.20

**Status Reporting System ..... 5.22**

Overview of Status Registers ..... 5.22

Structure of an SCPI Status Register ..... 5.23

Contents of the Status Registers ..... 5.25

STB and SRE ..... 5.26
IST Flag and PPE ..... 5.26

ESR and ESE ....5.26
STATus:OPERation ....5.27
STATus:QUEStionable ....5.27
STATus:QUEStionable:LIMit1<1|2> ....5.27
STATus:QUEStionable:INTegrity... ....5.28
Application of the Status Reporting System ....5.29
Service Request ....5.29
Serial Poll ....5.30
Parallel Poll ....5.31
Query of an Instrument Status ....5.31
Error Queue ....5.32
Reset Values of the Status Reporting System ....5.32

# 5 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, 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 the *SCPI Reference* chapter.

## Special Terms and Notation

This section explains the meaning of special syntax elements used in the SCPI command reference sections. A general description of the SCPI command syntax can be found in section *SCPI Command Structure and Syntax* in Chapter *Remote Control*.

***ZVR compatibility***
*The SCPI command set for the R&S ZVT vector network analyzer has been designed for compatibility with older R&S network analyzers of the ZVR type. A special class of commands, marked with the symbol ZVR, has been implemented primarily for compatibility reasons. Use these commands if you want to maintain compatibility with ZVR analyzers. If you want to make full use of the ZVT features and don't need ZVR compatibility, you should use the generalized, R&S ZVT-specific commands. A link to a generalized command is provided in the description of each ZVR-specific command.*

- **Information in the command tables**

All commands are described according to the same scheme. The following information is provided:

1. Complete command syntax and parameter list
2. Description of the command and its relationship with other commands
3. List and description of the parameters with their numerical ranges, default values and default units
4. SCPI conformance information, supported command types (setting command, query)
5. Program example

- **Order of commands**

The commands are arranged in alphabetical order. SCPI systems or subsystems are arranged in one topic.

- **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).

- ***<Par_Name>*** In the command tables and lists, parameters are generally described by a name (literal) written in angle brackets (<>). This literal merely serves as a parameters description; in an application program it must be replaced by one of the possible settings reported in the detailed parameter description.
- **Example:** CONTrol:AUXiliary:C[:DATA] <numeric_value> with <numeric_value> = 0 to 15 possible command syntax: CONT:AUX:C 1
- **NAN** (Not A Number) 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.
- Upper/lower case

Upper/lower case characters characterize the long and short form of the mnemonics in a command. The short form consists of all upper-case characters, the long form of all upper case plus all lower case characters. On the ZVT, either the short form or the long form are allowed; mixed forms will generally not be recognized. Either the short form or the long form are permissible. The instrument itself does not distinguish upper case and lower case characters.

- S**pecial characters**
- **|** A vertical stroke in the parameter list characterizes alternative parameter settings. Only one of the parameters separated by | must be selected.
- **Example:** The following command has two alternative settings: `FORMat[:DATA] ASCii | REAL`
- **[ ]**Key words in square brackets can be omitted when composing the command header (see *SCPI Command Structure and Syntax*). 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 entered in the command or omitted.
- **{ }** Braces or curly brackets enclose one or more parameters that may be included zero or more times.
- **Numeric suffixes**

Symbols in angular brackets (**<Ch>, <Ch/Tr>, <Mk>,...**) denote 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 marker suffix must be in the range between 1 and 10, the number of ports depends on the analyzer model. No restrictions apply to channel, trace, and diagram area suffixes.

In remote control, one active trace can be selected for each channel; see *Active Traces in Remote Control*. This concept simplifies the remote control command syntax, because it allows the active trace in a particular channel to be referenced by means of the channel suffix. To keep the syntax transparent, <Ch> is used for channel settings (it denotes the configured channel), whereas <Ch/Tr> is used for trace settings (it denotes the active trace in the channel).

# Remote Control: Introduction

The instrument is equipped with different interfaces for remote control:

- A GPIB bus interface according to standard IEC 625.1/IEEE 488.1. The GPIB bus connector for direct connection to a controller is located on the rear panel of the instrument.
- Instruments connected to a Local Area Network can be controlled via the RSIB or VX 11 interface. Two connectors for LAN connection are located on the rear panel.

The *GPIB Explorer* is a easy-to-use program tool that allows you to obtain an overview of all implemented remote control programs, test programs, compile and run test scripts.

This section assumes basic knowledge of GPIB bus programming and operation of the controller. A description of the interface commands can be found in sections *GPIB Bus Interface* and *RSIB Interface Functions*, respectively.

SCPI compatibility

The analyzers are compatible to the final SCPI version 1999.0. Not all of 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 device-specific 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. Tables provide a fast overview of the bit assignment in the status registers. The tables are supplemented by a comprehensive description of the status registers.

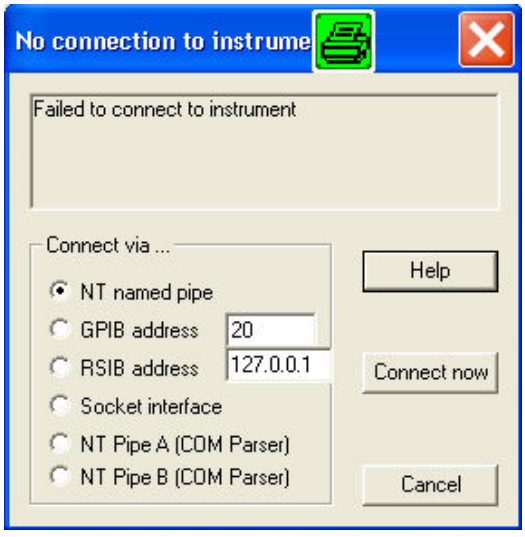
Reset values

In contrast to instruments with manual control, which are designed for maximum possible 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.

## GPIB Explorer

The *GPIB Explorer* is a easy-to-use program tool that allows you to obtain an overview of all implemented remote control programs, test programs, compile and run test scripts. The program can be opened from Windows XP's start menu: *Programs – R&S Network Analyzer – GPIB Explorer* or by starting the executable file `iecwin32.exe` in the program directory of the network analyzer (e.g. `C:\Program Files\Rohde&Schwarz\Network Analyzer\Bin`).

After the *GPIB Explorer* is started, the interface for the connection to the instrument can be selected in a dialog:

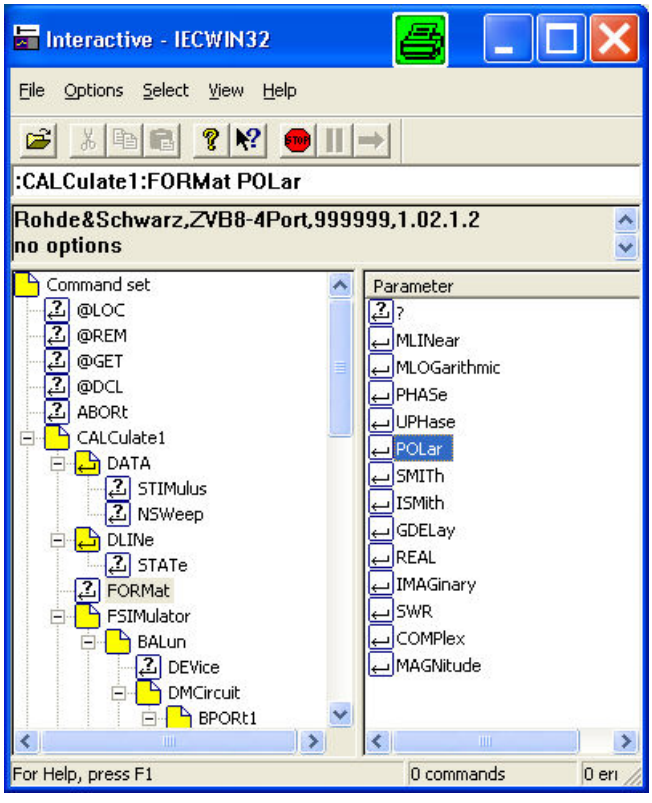



The following options are provided:

- NT named pipe (not supported at present)
- GPIB address (for connection to controllers equipped with a National Instrument GPIB interface using the GPIB bus connector)
- RSIB address (for LAN connection, requires an appropriate IP or local host address; see *LAN Connection* sections in Chapter 1)
- Socket interface (not supported at present)
- Named pipe A/B (only for local connection on the analyzer, recommended for remote test on the instrument)

Select *Info* – *Setup Info* to look up the IP address information of your analyzer.

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.




Refer to the GPIB explorer's help system for more information.

## 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 as soon as it receives a command from the controller. If the instrument is controlled via RSIB or VX 11 interface, the alternative commands *@REM* and *@LOC* can be used to switch from manual to remote control and back.

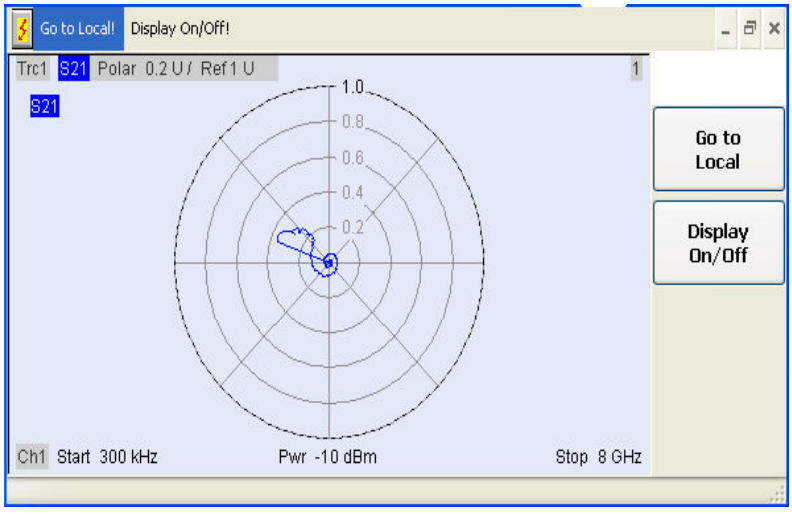
While remote control is active, operation via the front panel is disabled with the exception of two softkeys. The instrument settings are optimized for maximum measurement speed; the display is switched off:

The two softkeys in the remote screen are used to modify or quit the remote state:

- *Go to Local* switches the instrument to local state.
- *Display On/Off* switches the display on or off.

*Display On/Off* states

Switching on the display means that the analyzer shows the measurement screen with the current setups, 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 this is triggered by a remote control command; see `SYSTem:DISPlay:UPDate ONCE`.

You can create you own keys to replace the two default softkeys *Go to Local* and *Display On/Off.* See *Combining Manual and Remote Control.* A tooltip across the bottom of the remote screen indicates a remote command error, e.g. Remote Error : -222,"Data out of range;FREQ:STAR 1". You can switch off this tooltip using `SYSTem:ERRor:DISPlay OFF`.

Remote control: *@REM*
SYSTem:DISPlay:UPDate
SYSTem:USER:DISPlay:TITLe (define a title for the remote display)
SYSTem:ERRor:DISPlay (switch tooltip on or off)

## 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* menu or via GPIB bus. For remote control, addresses 0 through 30 are permissible. The GPIB address is maintained after a reset of the instrument settings.

Remote control: SYSTem:COMMunicate:GPIB[:SELF]:ADDRess

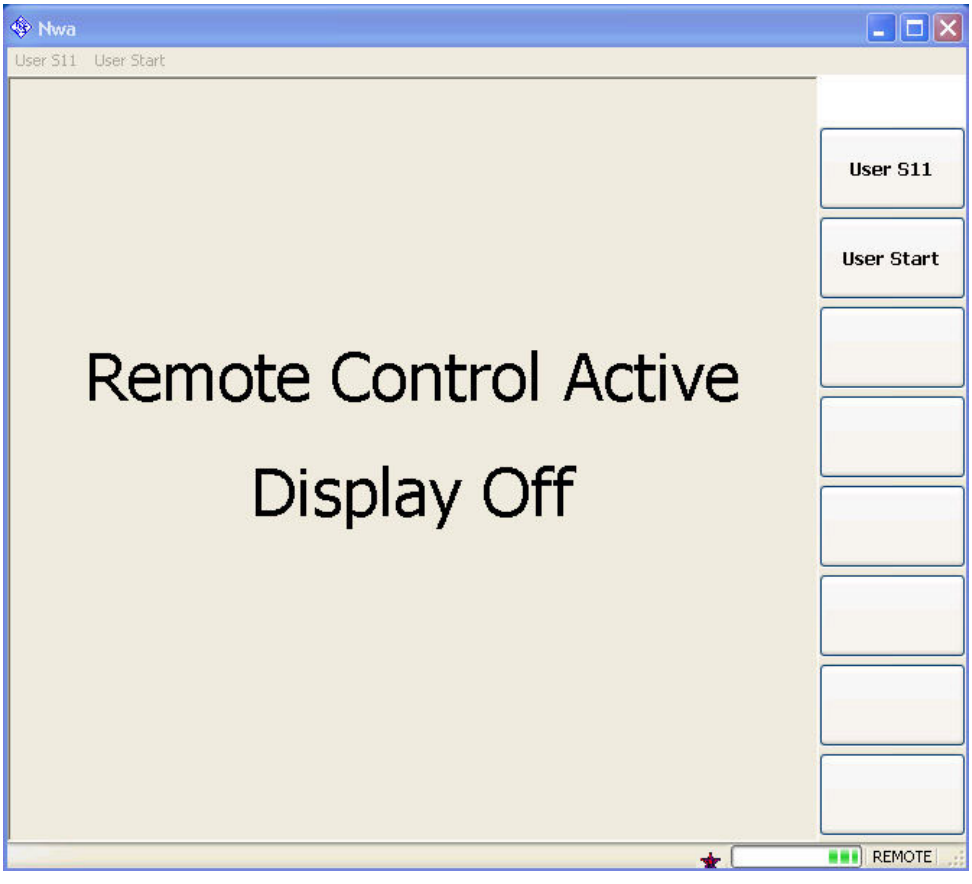
## Combining Manual and Remote Control

Using a remote control script is the quickest and easiest way of performing complicated tasks which need to be repeated many times. On the other hand, it is often preferable to control a previously configured measurement manually in order to observe the result on the screen.

The analyzer provides a number of 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 replace the default softkeys *Go to Local* and *Display On/Off.*




Pressing a user softkey or clicking the corresponding command across the top of the remote screen executes the assigned functionality.

If you want to use more than 8 user keys, you can easily introduce additional levels: Simply reserve one key for re-defining the entire user key bar. If you press or click the key, your remote script should re-assign the labels and functions of the existing keys. If you use this procedure repeatedly, you can emulate an arbitrary number of manual control features on your remote screen.

- **Menu key commands**

The commands in the `DISPlay:MENU:KEY...` subsystem open the manual control screen, activate menus or submenus for manual control, or execute a specified menu command.

DISPlay:MENU:KEY:EXECute '<menu_key>' can be used in different ways:

- If the menu key causes an event (e.g. selection of the measured quantity: S11, S12 etc.), the function is executed immediately. No further action is required; the instrument remains in remote control mode.
- If the menu key requires a numeric entry (e.g. entry of the sweep range: Start, Stop etc.) the command opens the numeric entry bar. The instrument is switched back to remote control mode as soon as the entry has been made manually.
- If the menu key requires several entries to be made in a dialog or in a wizard (e.g. the S-Param Wizard), the command opens this dialog/wizard. All entries can be made manually. The instrument is switched back to remote control mode as soon as the dialog or wizard is closed.

`DISPlay:MENU:KEY:EXECute '<menu_key>'` works for all menu keys, even though the analyzer may not provide an equivalent remote control command.

Remote control: SYSTem:USER:KEY
SYSTem:USER:KEY:FUNCtion
DISPlay:MENU:KEY:EXECute
DISPlay:MENU:KEY:SELect

## Return to Manual Operation

Return to manual operation can be initiated via the front panel or via remote control.

- Manually: Press the *Local* softkey in the remote screen.
- Via GPIB bus: CALL IBLOC(device%)
- Via RSIB or VX 11 interface: *@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 this is not the case, the analyzer switches back to remote control immediately.

Returning to manual control by pressing the *Local* softkey can be disabled by the GPIB Local Lockout Message (LLO; see *GPIB Bus Interface, Universal Commands)* which is also included in the NI commands SetRWLS (Set Remote With Lockout State) or SendLLO. This prevents unintentional switch-over, i.e. return to manual control is possible via the GPIB bus only.

Returning to manual control via the front panel keys can be enabled again by deactivating the REN control line of the GPIB bus .

## Messages

The messages transferred on the data lines of the GPIB bus or via the RSIB interface can be either interface messages or device messages.

### GPIB Bus Interface Messages

Interface messages are transferred on the data lines of the GPIB bus, the ATN control line being active. They are used for communication between controller and instrument and can only be sent by a computer which has the function of an GPIB bus controller.

Interface messages can be further subdivided into

- Universal commands act on all devices connected to the GPIB bus without previous addressing.
- Addressed commands only act on devices previously addressed as listeners.

The interface messages relevant to the instrument are listed in the GPIB bus section.

### RSIB Interface Messages

The RSIB interface allows the instrument to be controlled in a Local Area Network. The interface functions are matched to the function interface for GPIB bus programming from National Instruments.

### Device Messages (Commands and Device Responses)

Device messages are transferred via the data lines of the GPIB bus, the "ATN" control line not being active. The ASCII character set is used. A distinction is made according to the direction in which device messages are transferred:

- Commands are messages the controller sends to the instrument. They operate the device functions and request information.
- Device responses are messages 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 exactly defined in standard IEEE 488.2. Typical tasks are the management of the standardized status registers, reset and selftest.
   - Device-specific commands are functions that depend on the features of the instrument such as frequency settings. A majority of these commands has 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.

## SCPI Command Structure and Syntax

SCPI commands consist of a so-called header and, in most cases, 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 may consist of several key words. Queries are formed by directly appending a question mark to the header.

Common commands and device-specific commands differ in their syntax.

## 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.

## Device-Specific Commands

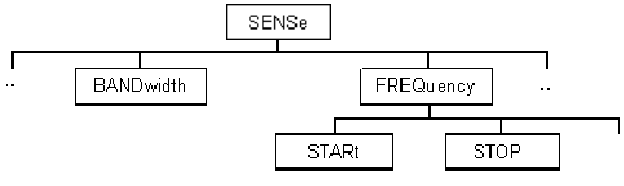
Device-specific commands are of hierarchical structure. The different levels are represented by combined headers. Headers of the highest level (root level) have only one key word. This key word denotes a complete command system.

**Example:**
SENSe This keyword denotes 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 key words being separated by a colon ":".

**Example:**
SENSe:FREQuency:STARt 1GHZ

This command is located 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 keywords

Some keywords 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 lGHZ

This command contains the key word SOURce in the first command level. It defines the frequency for sweep types operating at fixed frequency.

TRIGger:SOURce EXTernal

This command contains the key word SOURce in the second command level. It defines the trigger source “external trigger”.

- Optional keywords

Some command systems permit certain keywords to be optionally inserted into the header or omitted. These keywords 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 keywords.

**Example:**
TRIGger[:SEQuence]:SOURce EXTernal

This command defines the trigger source “external trigger”. The following command has the same effect:

TRIGger:SOURce EXTernal

*An optional keyword must not be omitted if its effect is additionally specified by a numeric suffix.*

- Long and short form

The key words feature a long form and a short form. Either the short form or the long form can be entered; other abbreviations are not permitted.

**Example:**
TRIGger:SOURce EXTernal
TRIG:SOUR EXT

*The short form is marked by upper case letters, the long form corresponds to the complete word. Upper case and lower case 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 types of parameter, 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.

## Structure of a Command Line

A command line may 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. Visual BASIC automatically produces 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:**
CALL IBWRT(device%, "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:** CALL IBWRT(device%, "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:

CALL IBWRT(device%, "`TRIG:SOUR EXT; TIM 0.1`")

However, a new command line always begins with the complete path.

Example: CALL IBWRT(device%, "TRIG:SOUR EXT ")
CALL IBWRT(device%, "TRIG:THR LOW ")

## 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`

# SCPI Parameters

Most commands require a parameter to be specified. 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.

## 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 may 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. In the case of 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.

## 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`

## Text Parameters

Text parameters observe the syntax rules for key words, 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. In the case of a query, the short form of the text is provided.

**Example:** Setting command: `TRIGger:SOURce EXTernal`
Query: `TRIGger:SOURce?` returns `EXT`

## 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'

## Block Data Format

Block data are 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.

## Overview of Syntax Elements

: The colon separates the key words 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.

# 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 setups, 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. In 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.

## 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:

```
CALC4:PAR:SDEF 'Ch4Tr1', 'S11'
```

Create channel 4 (channel suffix 4) and a trace named *Ch4Tr1* to measure the input reflection coefficient $S_{11}$. The trace is created but not displayed.

```
DISP:WIND2:STAT ON
```

Create diagram area no. 2 (window suffix 2).

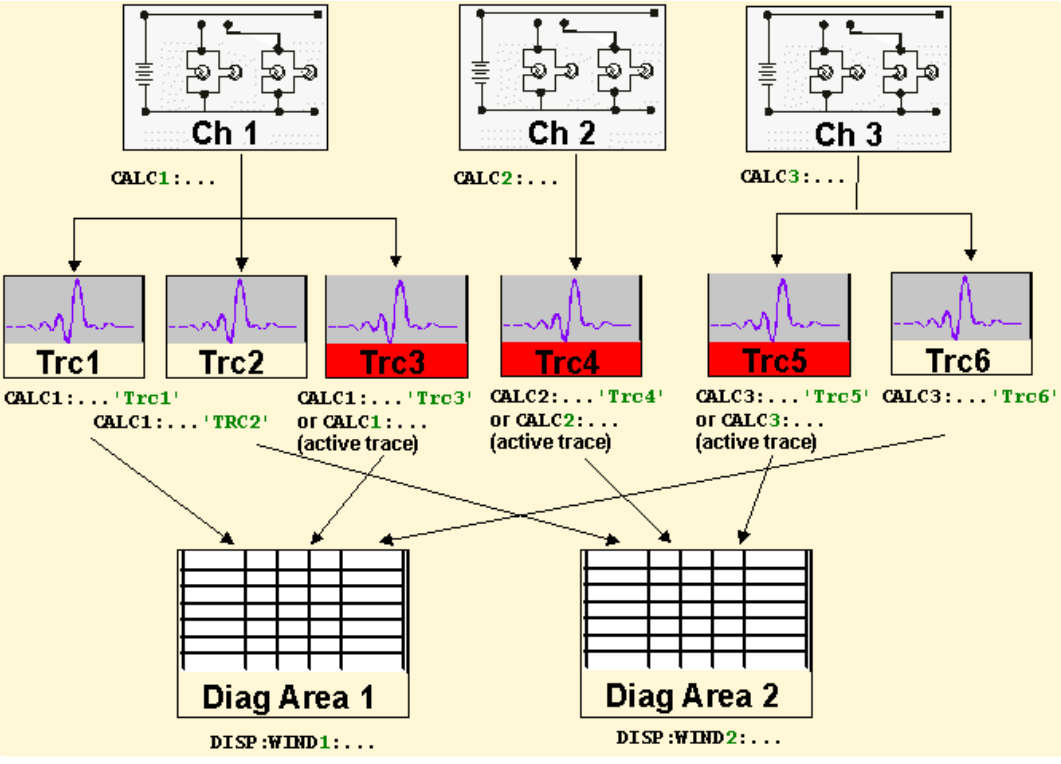
```
DISP:WIND2:TRAC9:FEED 'CH4TR1'
```

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.

## 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 *Trace Settings* in Chapter 3.

In remote control, each channel contains an active trace (unless the channel contains no trace at all), so the notion of *active channel* is meaningless. This principle actually simplifies the remote control command syntax, because it allows the active trace in a particular channel to be referenced by means of 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, click 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 may be different.

The following program example illustrates how to create, select and reference traces. It is instructive to observe the analyzer screen in order to check the effect of each step.

```
*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 new trace named *Trc2,* assigned to channel no. 1 (the suffix 1 after CALC, may be omitted), and display the trace. The new trace automatically becomes the active trace for manual and for remote control. To check this, click *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
```

To verify that *Trc2* is also active for remote control, use the channel suffix 1 after CALC (may be omitted) to reference the active trace in channel 1 and create a marker *Mkr 1*. The marker is assigned to *Trc2*.

```
CALC:PAR:SEL 'Trc1'; CALC1:MARK ON
```

Select the old default trace *Trc1* as the active trace for remote control. Create a new 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 denotes the active trace in the channel).*

## 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 in order 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 *Typical Stages of a Measurement*):

- 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.

```
*RST; INITiate:CONTinous OFF
```

Activate single sweep mode for all channels (including the channels created later).

```
INITiate:SCOPe SING
```

State that a single sweep will be performed in the active channel only.

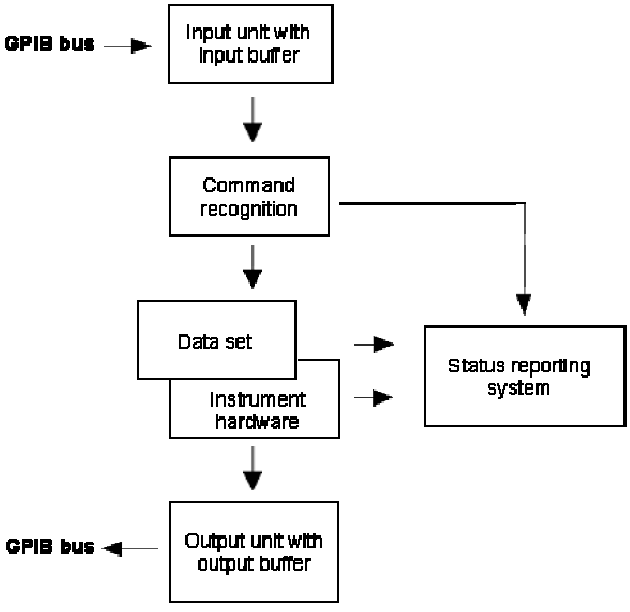
```
INITiate[:IMMediate]; *WAI
```

Start a single sweep in channel no. 2, wait until the sweep is terminated before proceeding to the next command (see *Command Synchronization*).

# Command Processing

Command ProcessingThe block diagram below shows how GPIB bus commands are serviced in the instrument.

The individual components work independently and simultaneously. They communicate with each other by means of so-called "messages".




## Input Unit

The input unit receives commands character by character from the GPIB bus and collects them in the input buffer. The input unit sends a message to the command recognition as soon as the input buffer is full or as soon as 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 GPIB bus traffic is stopped and the data received up to then is processed. Subsequently the GPIB bus 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.

## 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 ignored.

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").

## Data Set 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 set is a detailed software reproduction of the instrument hardware.

GPIB bus 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. As this is only ever effected at the end of a command line, 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 detection is made that execution is not possible, an "execution error" is signalled 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 FM and PM). 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 (cf. section *STATus:OPERation Register*). The hardware executes the settings and resets the bit again as soon as the new state has settled. This fact can be used to synchronize command servicing.

GPIB bus queries induce the data set management to send the desired data to the output unit.

## 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 Status Reporting System.

## 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, the controller waits until it has reached its time limit. This behavior is specified by SCPI.

## Command Sequence and Command Synchronization

What was said above makes clear that overlapping execution is possible in principle for all commands. Equally, setting commands within one command line are not absolutely serviced in the order in which

they have been received.

In order to make sure that commands are actually carried out in a certain order, each command must be sent in a separate command line, that is to say, with a separate IBWRT()–call.

The analyzer provides only one command, `INITiate<Ch>[:IMMediate]`, that has been implemented for overlapped execution. What is said below is not relevant for the other (sequential) SCPI commands.

Preventing overlapping execution

In order to prevent an overlapping execution of commands, one of commands *OPC, *OPC? or *WAI must be used. All three commands cause a certain action only to be carried out after the hardware has been set and has settled. By suitable programming, the controller can be forced to wait for the respective action to occur (see table below).

| Command | Action after the hardware has settled | Programming the controller |
|---|---|---|
| *OPC | Setting the operation complete bit in the ESR | – Setting bit 0 in the ESE<br>– Setting bit 5 in the SRE<br>– Waiting for service request (SRQ) |
| *OPC? | Writing a "1" into the output buffer | Addressing the instrument as a talker |
| *WAI | Executing the next command<br>Note: The GPIB bus handshake is not stopped | Sending the next command |

# 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 via GPIB bus (STATus... commands).

Hierarchy of status registers

As shown in the graphical overview, the status information is of hierarchical structure.

- STB, SRE The register STatus Byte (STB) and its associated mask register Service Request Enable (SRE) form 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.
- **ESR, SCPI registers**

The STB receives its information from the following registers:

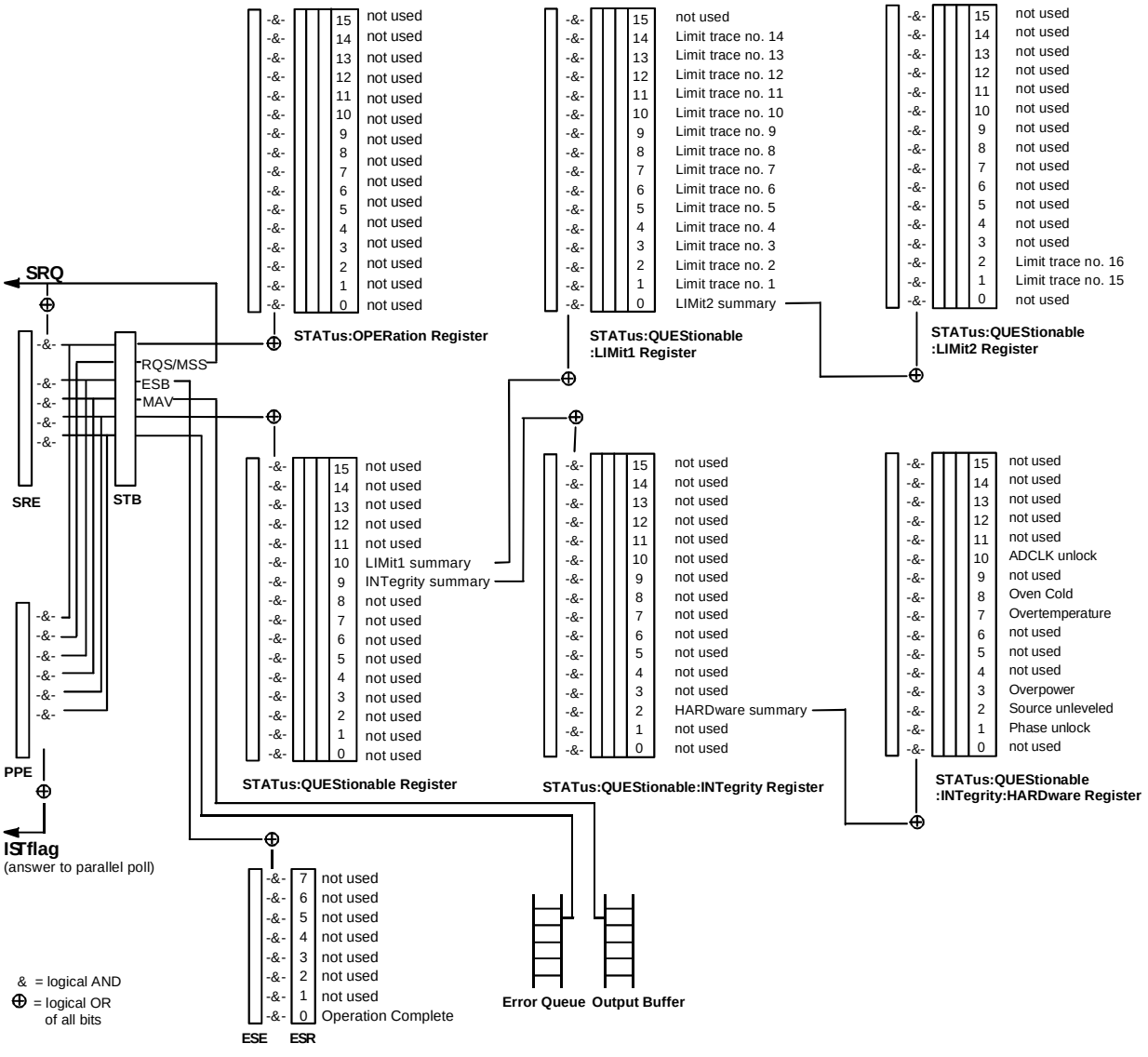
- The Event Status Register (ESR) with the associated mask register standard event status enable (ESE).
- The STATus:OPERation and STATus:QUEStionable registers which are defined by SCPI and contain detailed information on the instrument.
- IST, PPE The IST flag ("Individual STatus"), like the SRQ, combines 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 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 and thus is represented in the overview.

All status registers have the same internal structure.

*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.*

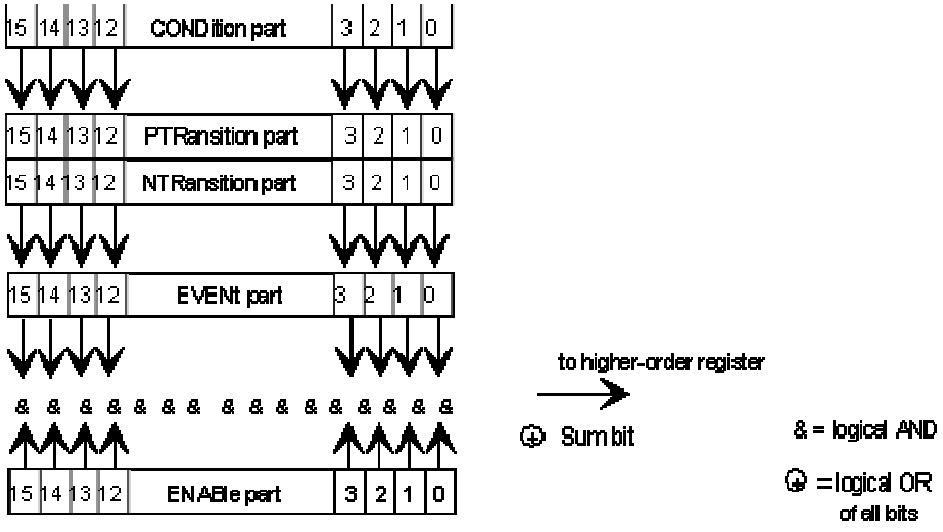
## Overview of Status Registers

The status registers of the network analyzer are implemented as shown below.




## Structure of an 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.




Description of the five status register parts

The five parts of an SCPI register have different properties and function:

- **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 nondestructive.

- **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.

As shown in the graphical overview, the status information is of hierarchical structure.

- STB, SRE The register STatus Byte (STB) defined in IEEE 488.2 and its associated mask register Service Request Enable (SRE) form 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.
- **ESR, SCPI registers**

The STB receives its information from the following registers:

- The standard IEEE 488.2 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 IST flag ("Individual STatus"), 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 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 and thus is represented in the overview.

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.

## 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 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.

## 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.

SRE and Service Request

The STatus Byte (STB) is linked to the Service Request Enable (SRE) register on a bit-by-bit basis.

- The STB corresponds to the CONDition part of an SCPI register, indicating the current instrument state.
- The SRE corresponds to the ENABle part of an 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 on the GPIB bus.

Bit 6 of the SRE is ignored, because it corresponds to the summary bit of the STB.

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?*.

The bits in the STB are defined as follows:

| Bit No. | Meaning |
|---|---|
| 3 | **QUEStionable status summary bit**<br>This bit is set if an EVENt bit is set in the QUEStionable register and the associated ENABle bit is set to 1.<br>The bit indicates a questionable instrument status, which can be further pinned down by polling the QUEStionable register. |
| 5 | **ESB bit**<br>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.<br>Setting of this bit implies an error or an event which can be further pinned down by polling the event status register. |

## IST Flag and PPE

In analogy to the Service Request (SRQ), the IST flag combines the entire status information in a single bit. It can be queried by means of 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 the ORing of all results.

Related common commands

The IST flag is queried using the command **IST?*.

The PPE can be set using **PRE* and read using command **PRE?*.

## 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 an SCPI register, indicating the current instrument state.
- The ESE corresponds to the ENABle part of an 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 is set.

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?*.

The bits in the ESR are defined as follows:

| Bit No. | Meaning |
|---|---|
| 0 | **Operation Complete**<br>This bit is set on receipt of the command *OPC after all previous commands have been executed. |

## 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:

## STATus:QUEStionable

The STATus:QUEStionable register indicates whether the data currently being acquired is of questionable quality. It can be queried using the commands `STATus:QUEStionable:CONDition?` or `STATus:QUEStionable[:EVENt]?`

The bits in the STATus:QUEStionable register are defined as follows:

| Bit No. | Meaning |
|---|---|
| 10 | **Limit Register summary**<br>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:LIMit1<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**<br>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<br>This bit is set if any point on trace no. 1 fails the limit check. |
| ... | ... |

| | |
|---|---|
| 14 | **Failed Limit Check for Trace no. 14**<br>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<br>This bit is set if any point on trace no. 15 fails the limit check. |
| 2 | **Failed Limit Check for Trace no. 16**<br>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 *Error Messages* section 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**<br>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. |

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 | **Phase Unlock**<br>This bit is set if the source has lost phase lock, which is possibly caused by a reference channel open or a hardware failure. |

| | |
|---|---|
| 2 | **Unleveled**<br>This bit is set if the source power is unleveled. |
| 3 | **Overpower**<br>This bit is set if too much power is detected at the input. This is from either using an amplifier, or a hardware failure.<br> <br>***The internal source power is automatically switched off when too much power is detected at the input. Correct your test setup, then switch on the internal source using*** ***`DIAGnostic:SERVice:RFPower ON`.*** |
| ... | **Not used** |
| 7 | **Overtemperature**<br>This bit is set if the source temperature sensor exceeds the limit. It could result from restricted airflow or a broken fan. |
| 8 | **Oven Cold**<br>This bit is set if the oven for the optional oven quartz (OCXO, 10 MHz reference oscillator, option ZVAB-B4, order no. 1164.1757.02) is not at its operating temperature. Possibly the oven is broken. |
| 9 | **Not used** |
| 9 | **ADCLK unlock**<br>This bit is set in case of a AD converter clock frequency lock failure. |
| ... | **Not used** |

# 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 are used:

- Service request (SRQ) initiated by the measuring device
- Serial poll of all devices in the bus system, initiated by the controller in order to find out who sent a SRQ and why
- Parallel poll of all devices
- Query of a specific instrument status by means of commands
- Query of the error queue

## Service Request

The measuring device 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 the graphical overview, 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 should be set to "1".

Examples:

- **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).

As soon as all commands preceding *OPC have been completed, the instrument generates an SRQ.

- **Generate an SRQ when a limit is exceeded**

4. Set bit 3 in the SRE (summary bit of the STATus:QUEStionable register, set after STATus:PRESet)
5. Set bit 10 in the STATus:QUEStionable:ENABle register (summary bit of the STATus:QUEStionable:LIMit1 register)
6. Set bit 1 in the STATus:QUEStionable:LIMit1:ENABle register

The analyzer generates a 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.

- **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:

7. STB? (query the contents of the status byte in decimal form)

If bit 3 (QUEStionable summary bit) is set, then:

8. STAT:QUES:EVENT? (query STATus:QUEStionable register)

If bit 10 (QUEStionable:LIMit1 summary bit) is set, then:

9. Query STAT:QUES:LIMit1:EVENT? (query STATus:QUEStionable:LIMit1 register)

If bit 1 is set, then the first trace failed the limit check.

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.*

## 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 by means of *STB?.

Serial poll procedure

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 quick BASIC command for executing a serial poll is "IBRSP()".

The serial poll is mainly used to obtain a fast overview of the state of several instruments connected to the GPIB bus.

## Parallel Poll

In a parallel poll, up to eight instruments are simultaneously requested by the controller by means of 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".

Parallel poll procedure

In addition to the SRE register, which determines the conditions under which an SRQ is generated, there is a Parallel Poll Enable register (PPE) which is ANDed with the STB bit by bit, considering bit 6 – AND as well. 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 by means of the command "*IST?".

The instrument first has to be set for the parallel poll using the quick BASIC command "IBPPC()". This command allocates a data line to the instrument and determines whether the response is to be inverted. The parallel poll itself is executed using "IBRPP()".

The parallel poll method is mainly used to find out quickly which one of the instruments connected to the GPIB bus has sent a service request. To this effect, SRE and PPE must be set to the same value.

## Query of an Instrument Status

Each part of any status register can be read by means of 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 (STATus:QUEStionable...)

All queries return a decimal number which represents the bit pattern of the status register. This number is evaluated by the controller program.

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 STatus Byte) are set.

Queries are usually used after an SRQ in order to obtain more detailed information on the cause of the SRQ.

### Error Queue

Each error state in the instrument leads to an entry in the error queue. The entries of the error queue are detailed plain text error messages that can be looked up in the *Error Log* or queried via the GPIB bus 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 any more, 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.

## Reset Values of the Status Reporting System

The table below indicates the effects of various commands upon the status reporting system of the analyzer.

| **Event** | **Switching on supply voltage Power-On-Status-Clear** | | **DCL,SDC(Device Clear, Selected Device Clear)** | ***RST or SYSTem:PRESet** | **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,<br>Fill ENABle parts of all other registers with "1". | | yes | | | yes | |
| Fill PTRansition parts with „1"<br>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.

# Contents of Chapter 6


## 6 Command Reference ..... 6.1

**Special Terms and Notation ..... 6.1**

**Common Commands ..... 6.3**

**SCPI Command Reference ..... 6.5**

CALCulate ..... 6.5

CONFigure... ..... 6.83

CONTrol... ..... 6.87

DIAGnostic:SERVice ..... 6.89

DISPlay... ..... 6.90

FORMat... ..... 6.105

HCOPy... ..... 6.106

HCOPy:DESTination <string> | 'MMEM' ..... 6.106

INITiate<Ch>... ..... 6.111

INPut... ..... 6.112

INSTrument... ..... 6.113

MEMory... ..... 6.114

MMEMory... ..... 6.115

OUTPut<Pt>... ..... 6.133

PROGram ..... 6.136

[SENSe...] ..... 6.137

SOURce<Ch>:... ..... 6.229

SOURce<Ch>:POWer... ..... 6.235

SOURce<Ch>:POWer:CORRection... ..... 6.241

SOURce<Ch>:TDIF... ..... 6.252

STATus... ..... 6.256

SYSTem... ..... 6.263

TRACe ..... 6.275

TRIGger<Ch>... ..... 6.279

# 6 Command Reference

This chapter lists all common commands and SCPI commands implemented by the analyzer.

## Special Terms and Notation

This section explains the meaning of special syntax elements used in the SCPI command reference sections. A general description of the SCPI command syntax can be found in section *SCPI Command Structure and Syntax* in Chapter *Remote Control*.

***Compatibility with ZVR and other instruments***
*The SCPI command set for the R&S ZVA vector network analyzer has been designed for compatibility with older R&S network analyzers of the ZVR type. A special class of commands, marked with the symbol* ZVR *, has been implemented primarily for compatibility reasons. Use these commands if you want to maintain compatibility with ZVR analyzers. If you want to make full use of the ZVA features and don't need ZVR compatibility, you should use the generalized, R&S ZVA-specific commands. A link to a generalized command is provided in the description of each ZVR-specific command.*
*Similar considerations also hold for other commands that have been "implemented for compatibility reasons".*

- **Information in the command tables**

  All commands are described according to the same scheme. The following information is provided:

  1. Complete command syntax and parameter list
  2. Description of the command and its relationship with other commands
  3. List and description of the parameters with their numerical ranges, default values and default units
  4. SCPI conformance information, supported command types (setting command, query)
  5. Program example

- **Order of commands**

  The commands are arranged in alphabetical order. SCPI systems or subsystems are arranged in one topic.

- **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).

  - ***<Par_Name>*** In the command tables and lists, parameters are generally described by a name (literal) written in angle brackets (<>). This literal merely serves as a parameters description; in an application program it must be replaced by one of the possible settings reported in the detailed parameter description.

- **Example:** CONTrol:AUXiliary:C[:DATA] <numeric_value>
  with <numeric_value> = 0 to 15
  possible command syntax: CONT:AUX:C 1

  - **NAN** (Not A Number) 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.
- **Upper/lower case**

  Upper/lower case characters characterize the long and short form of the mnemonics in a command. The short form consists of all upper-case characters, the long form of all upper case plus all lower case characters. On the ZVA, either the short form or the long form are allowed; mixed forms will generally not be recognized. The instrument itself does not distinguish upper case and lower case characters.
- **Special characters**
  - **|** A vertical stroke in the parameter list characterizes alternative parameter settings. Only one of the parameters separated by | must be selected.

- **Example:** The following command has two alternative settings:
  `FORMat[:DATA] ASCii | REAL`
  - **[ ]**Key words in square brackets can be omitted when composing the command header (see *SCPI Command Structure and Syntax*). 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 entered in the command or omitted.
  - **{ }** Braces or curly brackets enclose one or more parameters that may be included zero or more times.
- **Numeric suffixes**

  Symbols in angular brackets (**<Ch>, <Chn>, <Mk>,...**) denote 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 marker suffix must be in the range between 1 and 10, the number of ports depends on the analyzer model. No restrictions apply to channel, trace, and diagram area suffixes.

  In remote control, one active trace can be selected for each channel; see *Active Traces in Remote Control*. This concept simplifies the remote control command syntax, because it allows the active trace in a particular channel to be referenced by means of the channel suffix. To keep the syntax transparent, <Ch> is used for channel settings (it denotes the configured channel), whereas <Chn> is used for trace settings (it denotes the active trace in the channel).

# Common Commands

Common commands are described in the IEEE 488.2 (IEC 625-2) standard. These commands have the same effect on different devices. The headers of these commands consist of "*" followed by three letters. Many common commands are related to the status reporting system.

| Command | Parameters | Short Description |
|---|---|---|
| | | |
| ***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. | | Clear Status; no query |
| ***ESE Event Status Enable** sets the event status enable register to the value indicated. The query *ESE? returns the contents of the event status enable register in decimal form. | 0...255 | Event Status Enable |
| ***ESR? Event Status Read** returns the contents of the event status register in decimal form (0 to 255) and subsequently sets the register to zero. | | Standard Event Status Query; query only |
| ***IDN? IDeNtification query** returns the instrument identification. The response is of the form "Rohde&Schwarz,ZVA8-4Port,12345,0.10.1.23", where<br>ZVxx-nPort is the analyzer type<br>12345 is the serial number of the analyzer<br>0.10.1.23 is the firmware version number.<br>The ID string can be changed in the *System Config* dialog. | | Identification Query; query only |
| ***IST? Individual STatus query** returns the contents of the IST flag in decimal form (0 \| 1). The IST-flag is the status bit which is sent during a parallel poll. | | Individual Status Query; 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.<br>The query form writes a "1" into the output buffer as soon as all preceding commands have been executed. This is used for command synchronization. | | Operation Complete |
| ***OPT? OPTion identification query** queries the options included in the instrument and returns a list of the options installed. 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. | | Option Identification Query; query only |
| ***PCB Pass Control Back** indicates the controller address to which GPIB bus control is returned after termination of the triggered action. | 0...30 | Pass Control Back; no query |
| ***PRE Parallel poll Register Enable** sets parallel poll enable register to the value indicated. The query *PRE? returns the contents of the parallel poll enable register in decimal form. | 0...255 | Parallel Poll Register Enable |
| ***PSC Power on Status Clear** determines whether the contents of the ENABle registers is maintained or reset when the analyzer is switched on. | 0 \| 1 | Power On Status Clear |

| | | |
|---|---|---|
| *PSC = 0 causes the contents of the status registers to be maintained. Thus a service request can be triggered on switching on in the case of a corresponding configuration of status registers ESE and SRE.<br>*PSC ≠ 0 resets the registers. | | |
| ***RST ReSeT** sets the instrument to a defined default status. The command resets all setups, restoring the factory default values defined for remote control operation. In contrast the command `SYSTem:PRESet` resets the active setup only. The default settings are indicated in the description of commands. | | Reset;<br>no query |
| ***SRE 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. | 0...255 | Service Request Enable |
| ***STB? STatus Byte query** reads the contents of the status byte in decimal form. | | Status Byte Query;<br>query only |
| ***TRG TRiGger** triggers all actions waiting for a trigger event. In particular *TRG generates a manual trigger signal *(Manual Trigger).* This common command complements the commands of the `TRIGger` subsystem. | | Trigger;<br>no query |
| ***TST? self TeST query** triggers selftests of the instrument and returns an error code in decimal form. | | Self Test Query;<br>query only |
| ***WAI WAIt to continue** prevents servicing of the subsequent commands until all preceding commands have been executed and all signals have settled (see also command synchronization and **OPC)*. | | Wait to continue;<br>no query |

# SCPI Command Reference

## CALCulate

### CALCulate<Chn>:CLIMits...

This subsystem controls the composite limit check.

### CALCulate<Chn>:CLIMits:FAIL?

Returns a 0 or 1, to indicate whether or not a composite limit check on several traces has failed.

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace. This suffix is not relevant because the command provides a summary of all individual limit fails. |
| **Response** | 0 \| 1 - 0 represents pass for all individual limit checks (also returned if no limit check is active at all), 1 means that the limit checks for one or more traces failed. |
| *RST value | 0 |
| **SCPI, Command Types** | Device-specific, query only. |
| **Example:** | `*RST; CALC:LIM:CONT 1 GHZ, 2 GHZ`<br>Define an upper limit line segment in the stimulus range between 1 GHz and 2 GHz, using default response values.<br>`CALC:LIM:STAT ON; FAIL?`<br>Switch the limit check on and query the result.<br>`CALC:CLIM:FAIL?`<br>Query the result for the composite limit check. As only one trace is tested, the response should be equal to the previous response.. |

### CALCulate<Chn>:DATA...

This subsystem provides access to the results of a measurement.

*The 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`).*

## CALCulate<Chn>:DATA FDATa | SDATa | MDATa | SCORr1 | ... | SCORr27

Reads the current response values of the active data trace or memory trace and reads or writes error terms.

**<Chn>** Channel number used to identify the active trace. If unspecified the numeric suffix is set to 1.

**Parameters** See list of parameters below.

Range [def. unit] The data format is parameter-dependent; see below. The unit is the default unit of the measured parameter; see `CALCulate<Ch>:PARameter:SDEFine`.

*RST value –

**SCPI, Command Types** Confirmed, command (for error terms) or query.

**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.

The following parameters are related to trace data (see also *Data Flow* diagram):

| | |
|---|---|
| `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. |

The following parameters denote the error terms generated during a calibration:

| 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) |

Use the generalized command [SENSe<Ch>:]CORRection:COLLect:CDATa to read or write error terms for arbitrary analyzer ports.

## CALCulate<Ch>:DATA:SGRoup? FDATa | SDATa | MDATa

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.

| | |
|---|---|
| **<Ch>** | Channel number of the previously defined S-parameter group. |
| **Parameters**<br>*RST value | Output format for the S-parameter data, see CALCulate<Chn>:DATA.<br>– |
| **SCPI, Command Types** | Device-specific, query only. The order of the returned S-parameters within the S-parameter group is fixed; see example. |
| **Example:** | See CALCulate<Ch>:PARameter:DEFine:SGRoup |

## CALCulate<Chn>:DATA:NSWeep? SDATa, <Trace_Hist_Count>

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.

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace. |
| **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). |
| **<Trace_Hist_Count>** | Number of sweep to be read. 1 denotes the last sweep acquired, 2 denotes the second-last and so forth. |
| Range [def. unit]<br>*RST value | 1 to sweep count defined via [SENSe<Ch>:]SWEep:COUNt [–]<br>– |
| **SCPI, Command Types** | Device-specific, query only |
| **Example (see also Sweep History):** | SWE:COUN 10<br>Define the number of sweeps (10) to be measured in single sweep mode.<br>INIT:CONT OFF; INIT<br>Activate single sweep mode and start a single sweep in channel no. 1.<br>CALC:DATA:NSW? SDAT,5<br>Query the results of the 5th sweep. |

## CALCulate<Chn>:DATA:STIMulus?

Reads the stimulus values of the active data or memory trace.

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace. |
| **Parameters** | – |
| Range [def. unit] | The data is transferred in the data format defined via FORMat[:DATA]. The unit is the default unit of the sweep variable (Hz or dBm or s)**.** |
| *RST value | – |
| **SCPI, Command Types** | Device-specific, query only |
| **Example:** | `See CALCulate<Chn>:DATA?` |

## CALCulate<Chn>:DLINe...

This subsystem controls the horizontal line used to mark and retrieve response values (display line).

## CALCulate<Chn>:DLINe <numeric_value>

Defines the position (response value) of the horizontal line.

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace. If unspecified the numeric suffix is set to 1. |
| **Parameters** | See list of parameters below. |
| Range [def. unit] | The data is transferred in the data format defined via FORMat[:DATA]. The unit is the default unit of the measured parameter; see `CALCulate<Ch>:PARameter:SDEFine`**.** |
| *RST value | – |
| **SCPI, Command Types** | Device-specific, command or query. |
| **Example:** | `*RST; CALC:DLIN 10` |
| | Define the position of the horizontal line in the default dB Mag diagram at +10 dBm. |
| | `CALC:DLIN:STAT ON` |
| | Display the defined horizontal line. |

## CALCulate<Chn>:DLINe:STATe <Boolean>

Switches the horizontal line on or off.

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace. |
| **<Boolean>** | ON \| OFF - Horizontal line on or off. |

| | |
|---|---|
| *RST value | OFF |
| **SCPI, Command Types** | Device-specific, command or query. |
| **Example:** | `See CALC:DLIN.` |

## CALCulate<Chn>:FILTer[:GATE]...

This subsystem defines the properties of the time gate optimize the time domain response (with option ZVAB-K2, *Time Domain)*.

## CALCulate<Chn>:FILTer[:GATE]:TIME:CENTer <numeric_value>

Defines the center time of the time gate.

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace |
| **<numeric_value>** | Center time of the time gate |
| Range [def. unit] | -99.8999999 s to +99.8999999 s. The increment (parameters UP or DOWN) is 0.1 ns. [s] |
| *RST value | 1.5E-009 s |
| **SCPI, Command Types** | Confirmed, command or query |
| **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. |

## CALCulate<Chn>:FILTer[:GATE]:TIME:DCHebyshev <numeric_value>

Sets the sideband suppression for the Dolph-Chebyshev time gate. The command is only available if a Dolph-Chebyshev time gate is active (`CALCulate<Chn>:FILTer[:GATE]:TIME:WINDow DCHebyshev`).

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace |
| **<numeric_value>** | Sideband suppression |
| Range [def. unit] | 10 dB to 120 dB. The increment (parameters UP or DOWN) is 10 dB. [dB] |
| *RST value | 32 dB |
| **SCPI, Command Types** | Device-specific, command or query |
| **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.

## CALCulate<Chn>:FILTer[:GATE]:TIME:SHAPe MAXimum | WIDE | NORMal | MINimum

Selects the time gate to be applied to the time domain transform.

Use the generalized command `CALCulate<Chn>:FILTer[:GATE]:TIME:WINDow` if you wish to select a Dolph-Chebychev time gate.

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace |
| **Parameters** | MINimum – Steepest edges (rectangle)<br>WIDE – Normal gate (Hann)<br>NORM – Steep edges (Hamming)<br>Maximum – Maximum flatness (Bohman) |
| *RST value | WIDE |
| **SCPI, Command Types** | Device-specific, command or query |
| **Example:** | `*RST; CALC:FILT:TIME:SHAP?` |
| | Reset the instrument and query the type of time gate used. The response is `WIDE`. |

## CALCulate<Chn>:FILTer[:GATE]:TIME:SHOW <Boolean>

Enables or disables permanent display of the gate limits.

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace |
| **<Boolean>** | ON – Time gate permanently displayed<br>OFF – Time gate hidden |
| *RST value | OFF [–] |
| **SCPI, Command Types** | Device-specific, command or query |
| **Example:** | `See CALCulate<Chn>:FILTer[:GATE]:TIME:CENTer` |

## CALCulate<Chn>:FILTer[:GATE]:TIME:SPAN 

Defines the span of the time gate.

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace |
| **** | Span of the time gate. |
| Range [def. unit] | 2E-012 s to 200 s. The increment (parameters UP or DOWN) is 0.1 ns. [s] |
| *RST value | 5E-009 s |

| | |
|---|---|
| **SCPI, Command Types** | Confirmed, command or query |
| **Example:** | `See CALCulate<Chn>:FILTer[:GATE]:TIME:CENTer` |

## CALCulate<Chn>:FILTer[:GATE]:TIME:STARt <start>

Defines the start time of the time gate.

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace |
| **<start>** | Start time of the time gate. |
| Range [def. unit] | –100 s to +99.999999999998 s. The increment (parameters UP or DOWN) is 0.1 ns. [s] |
| *RST value | –1E-009 s |
| **SCPI, Command Types** | Confirmed, command or query |
| **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. |

***Note:*** *If the start frequency entered is greater than the current stop frequency (`CALCulate<Chn>:FILTer[:GATE]:TIME:STOP`), the stop frequency is set to the start frequency plus the minimum frequency span (`CALCulate<Chn>:FILTer[:GATE]:TIME:SPAN`).*

## CALCulate<Chn>:FILTer[:GATE]:TIME:STATe <Boolean>

Determines whether the time gate for trace no. <Chn> is enabled.

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace |
| **<Boolean>** | ON – Time gate enabled<br>OFF – Time gate disabled |
| *RST value | OFF [–] |
| **SCPI, Command Types** | Confirmed, command or query |
| **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 is *0;0*. |

## CALCulate<Chn>:FILTer[:GATE]:TIME:STOP <numeric_value>

Defines the stop time of the time gate.

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace |
| **<numeric_value>** | Stop time of the time gate. |
| Range [def. unit] | –99.999999999998 s to +100 s. The increment (parameters UP or DOWN) is 0.1 ns. [s] |
| *RST value | +4E-009 s |
| **SCPI, Command Types** | Confirmed, command or query |
| **Example:** | `See CALCulate<Chn>:FILTer[:GATE]:TIME:STARt.` |

***Note:*** *If the stop frequency entered is smaller than the current start frequency (`CALCulate<Chn>:FILTer[:GATE]:TIME:STARt`), the start frequency is set to the stop frequency minus the minimum frequency span (`CALCulate<Chn>:FILTer[:GATE]:TIME:SPAN`).*

## CALCulate<Chn>:FILTer[:GATE]:TIME[:TYPE] BPASs | NOTCh

Selects the time gate filter type, defining what occurs to the data in the specific time region.

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace |
| **Parameters** | BPASs – Band pass filter: pass all information in specified time region and reject all else<br>NOTCh – Notch filter: reject all information in specified time region and pass all else. |
| *RST value | BPASs |
| **SCPI, Command Types** | Confirmed, command or query |
| **Example:** | `*RST; CALC:FILT:TIME:STAT ON` |
| | `Reset the instrument and enable the time gate.` |
| | `CALC:FILT:TIME NOTCh` |
| | Select a notch filter in order to reject unwanted pulses. |

## CALCulate<Chn>:FILTer[:GATE]:TIME:WINDow RECT | HAMMing | HANN | BOHMan | DCHebyshev

Selects the time gate to be applied to the time domain transform.

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace |
| **Parameters** | RECT – Steepest edges (rectangle)<br>HANN – Normal gate (Hann)<br>HAMMing – Steep edges (Hamming)<br>BOHMan – Minimum flatness (Bohman)<br>DCHebyshev – Arbitrary gate shape (Dolph-Chebychev) |
| *RST value | HANN |
| **SCPI, Command Types** | Confirmed (with some device-specific parameters), command or query |

**Example:**

```
See CALCulate<Chn>:FILTer[:GATE]:TIME:DCHebychev
```

## CALCulate<Chn>:FORMat...

This subsystem determines a post-processing of the measured data in order to obtain various display formats.

## CALCulate<Chn>:FORMat MLINear | MLOGarithmic | PHASe | UPHase | POLar | SMITh | ISMith | GDELay | REAL | IMAGinary | SWR | COMPlex | MAGNitude

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 Trace - Format and CALCulate<Chn>:PARameter commands. Nevertheless, it is advisable to check which display formats are generally appropriate for an analysis of a particular measured quantity; see Measured Quantities and Display Formats.*

**<Chn>** Channel number used to identify the active trace. If unspecified the numeric suffix is set to 1.

**Parameters** See list of parameters below.

*RST value MLOGarithmic

**SCPI, Command Types** Confirmed, command or query

**Example:**

```
CALC4:PAR:SDEF 'Ch4Tr1', 'S11'
```

Create channel 4 and a trace named *Ch4Tr1* to measure the input reflection coefficient S11. The trace becomes the active trace in channel 4.

```
CALC4:FORM MLIN; DISP:WIND:TRAC:FEED 'CH4TR1'
```

Calculate the magnitude of S11 and display it in a linearly scaled Cartesian diagram.

Assume that the result at a sweep point is given by the complex quantity z = x + jy. The meaning of the parameters is as follows (see also table in `CALCulate<Chn>:MARKer<Mk>:FORMat` description):

| | |
|---|---|
| `MLINear` | Calculate the magnitude of z, to be displayed in a Cartesian diagram with a linear scale |
| `MLOGarithmic` | Calculate the magnitude of z, displayed in a Cartesian diagram with a logarithmic scale |
| `PHASe` | Phase of z, displayed in a Cartesian diagram with a linear vertical axis |
| `UPHase` | Unwrapped phase of z, displayed in a Cartesian diagram with a linear vertical axis |
| `POLar` | Magnitude and phase, displayed in a polar diagram |
| `SMITh` | Magnitude and phase, displayed in a Smith chart |
| `ISMith` | Magnitude and phase, displayed in an inverted Smith chart |
| `GDELay` | Group delay, displayed in a Cartesian diagram |
| `REAL` | Real part (x), displayed in a Cartesian diagram |
| `IMAGinary` | Imaginary part (y), displayed in a Cartesian diagram |
| `SWR` | Standing Wave Ratio (SWR), displayed in a Cartesian diagram |
| ZVR `COMPlex` | x, y, displayed in a polar diagram |

| ZVR MAGNitude | Magnitude (sqrt($x^2$ + $y^2$)), displayed in a Cartesian diagram with a logarithmic scale |
|---|---|

## CALCulate<Chn>:FSIMulator...

This subsystem enables the virtual *SAW Matching Network* and defines its parameters. When the virtual network is enabled, port 1 serves as an unbalanced port, the physical ports no. 2 and 4 of the analyzer are combined to a logical (balanced) port no. 2.

NOTE *The commands in this subsystem are supported for the sake of compatibility with older firmware versions. Starting with firmware V1.70 the commands have been superseded by the command in the CALCulate:TRANsform:VNETwork... subsystem. Virtual network (de-)embedding is controlled by the Virtual Transform menu.*

## CALCulate<Ch>:FSIMulator:BALun:DEVice SBALanced

Selects the type of matching network to be simulated. The present firmware version provides a virtual matching network for DUTs with a single-ended and a balanced port.

TIP Use this command in order to avoid problems should the default value change in future firmware versions.

| | |
|---|---|
| **<Ch>** | Channel number. If unspecified the numeric suffix is set to 1. |
| **Parameters**<br>Range [def. unit]<br>*RST value | Matching network type.<br>SBALanced: single-ended/balanced [–].<br>SBALanced |
| **SCPI, Command Types** | Device-specific, command or query. |
| **Example:** | `See CALCulate<Ch>:FSIMulator:BALun:DMCircuit:BPORt1:PARameters:C` |

## CALCulate<Ch>:FSIMulator:BALun:DMCircuit:BPORt2:PARameters:C <Capacitance>

Defines the serial capacitance for the matching network.

| | |
|---|---|
| **<Ch>** | Channel number. |
| **<Capacitance>**<br>Range [def. unit]<br>*RST value | Serial capacitance for logical port no. 2.<br>–1 mF to 1 mF [F (farad)]. The increment (parameters UP or DOWN) is 1E-015 F.<br>1E-009 F |
| **SCPI, Command Types** | Device-specific, command or query. |
| **Example:** | `*RST; CALC:FSIM:BAL:DEV SBAL`<br>Select the matching network type *single-ended/balanced* (the default setting may change in future firmware versions).<br>`CALC:FSIM:STAT ON` |

Activate the matching network.

```
CALC:FSIM:BAL:DMC:BPOR2:PAR:C 2 NF; :L 2 NH
```

Set the serial capacitance to 2 nF and the parallel inductance to 2 nH.

```
:CALC:FSIM:BAL:DZC:BPOR2:ZCOM 50; ZDIF 150
```

Set the common mode impedance to 50 Ω, the differential mode impedance to 150 Ω.

## CALCulate<Ch>:FSIMulator:BALun:DMCircuit:BPORt2:PARameters:L <Inductance>

Defines the parallel inductance for the matching network.

| | |
|---|---|
| **<Ch>** | Channel number. |
| **<Inductance>** | Parallel inductance for logical port no. 2. The increment (parameters UP or DOWN) is 1E-015 H. |
| Range [def. unit] | –1 mH to 1 mH [H (henry)] |
| *RST value | 1E-009 H |
| **SCPI, Command Types** | Device-specific, command or query. |
| **Example:** | `See CALCulate<Ch>:FSIMulator:BALun:DMCircuit:BPORt1:PARameters:C` |

## CALCulate<Ch>:FSIMulator:BALun:DZConversion:BPORt2:ZCOMmon[:R] <Impedance>

Defines the common mode impedance for the balanced port no. 2.

| | |
|---|---|
| **<Ch>** | Channel number. |
| **<Impedance>** | Common mode impedance. The increment (parameters UP or DOWN) is 1 Ω. |
| Range [def. unit] | 0 Ω to 3.4028234664E+032 MΩ [Ω (Ohm)] |
| *RST value | 25 Ω |
| **SCPI, Command Types** | Device-specific, command or query. |
| **Example:** | `See CALCulate<Ch>:FSIMulator:BALun:DMCircuit:BPORt1:PARameters:C` |

## CALCulate<Ch>:FSIMulator:BALun:DZConversion:BPORt2:ZDIFferent[:R] <Impedance>

Defines the differential mode impedance for the balanced port no. 2.

| | |
|---|---|
| **<Ch>** | Channel number. |
| **<Impedance>** | Differential mode impedance. The increment (parameters UP or DOWN) is 1 Ω. |
| Range [def. unit] | 0 Ω to 3.4028234664E+032 MΩ [Ω (Ohm)] |

| | |
|---|---|
| *RST value | 100 Ω |
| **SCPI, Command Types** | Device-specific, command or query. |
| **Example:** | `See CALCulate<Ch>:FSIMulator:BALun:DMCircuit:BPORt1:PARameters:C` |

## CALCulate<Ch>:FSIMulator:STATe <Boolean>

Activates or deactivates the virtual matching network. When the network is activated, port 1 serves as an unbalanced port, the physical ports no. 2 and 4 of the analyzer are combined to a logical (balanced) port no. 2.

| | |
|---|---|
| **<Ch>** | Channel number. |
| **<Boolean>** | ON - Enable the virtual matching network and adjust the port configuration<br>OFF - Disable the virtual matching network |
| *RST value | OFF |
| **SCPI, Command Types** | Device-specific, command or query. |
| **Example:** | `See CALCulate<Ch>:FSIMulator:BALun:DMCircuit:BPORt1:PARameters:C.` |

## CALCulate<Chn>:LIMit...

This subsystem defines the limit lines and controls the limit check.

## CALCulate<Chn>:LIMit:CONTrol[:DATA] <numeric_value>,<numeric_value>{,<numeric_value>,<numeric_value>}

Defines the stimulus values of the limit line and/or creates new limit line segments.

**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]`).

*The generated segments are upper or lower limit line segments, depending on the CALCulate<Chn>:LIMit:SEGMent<Seg>:TYPE setting.*

*CALCulate<Chn>:LIMit:CONTrol[:DATA] does not overwrite the type setting.*

To define additional new limit line segments without overwriting the old segments use CALCulate<Chn>:LIMit:DATA.

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace. |
| **<numeric_value>** | Pair(s) of frequency, power or time values, to be defined in accordance with the domain setting (CALCulate<Chn>:LIMit:CONTrol:DOMain). |
| Range [def. unit] | Almost no restriction for limit segments; see *Rules for Limit Line Definition.* [Hz for frequency sweeps, dBm for power sweeps, s for time sweeps] |
| *RST value | –<br>A segment that is created implicitly, e.g. by means of CALCulate<Chn>:LIMit:UPPer[:DATA] or CALCulate<Chn>:LIMit:LOWer[:DATA,], covers the maximum sweep range of the analyzer. |
| **SCPI, Command Types** | Confirmed, with query. |
| **Example:** | `*RST; CALC:LIM:CONT 1 GHZ, 2 GHZ`<br>Define an upper limit line segment in the stimulus range between 1 GHz and 2 GHz, using default response values.<br>`CALC:LIM:DISP ON`<br>Show the limit line segment in the active diagram. |

## CALCulate<Chn>:LIMit:CONTrol:DOMain FLIN | FLOG | FSEG | FSINgle | TLIN | TLOG | PLIN | PLOG | PSINgle

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 are defined via CALCulate<Chn>:LIMit:RDOMain:...

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace. |
| **Parameters** | Keywords for the units of the stimulus values.<br>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. |
| Range [def. unit] | The parameters form three groups:<br>▪ FLIN, FLOG, FSEG, and FSINgle select frequency units [default: Hz] for the limit line.<br>▪ TLIN and TLOG select time units [default: s].<br>▪ PLIN, PLOG and PSINgle select power units [default: dBm]. |
| *RST value | FLIN |
| **SCPI, Command Types** | Device-specific, no query. |
| **Example:** | `SWE:TYPE POW`<br>Select a power sweep.<br>`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 DBM, -10 DBM
```

Define a limit line segment in the stimulus range between –20 dBm and –10 dBm.

## CALCulate<Chn>:LIMit:CONTrol:SHIFt <numeric_value>

Shifts the limit line in horizontal direction.

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace. |
| **<numeric_value>** | Amount that the limit line is shifted. |
| Range [def. unit] | Almost no restriction for limit segments; see *Rules for Limit Line Definition.* [Hz for frequency sweeps, dBm for power sweeps, s for time sweeps, see `CALCulate<Chn>:LIMit:CONTrol:DOMain`] |
| *RST value | – |
| **SCPI, Command Types** | Device-specific, no query. |

**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 extends from 1000000001 (Hz) to 2000000001 (Hz).

## CALCulate<Chn>:LIMit:DATA <type>, <start_stim>, <stop_stim>, <start_resp>, <stop_resp>{,<type>, <start_stim>, <stop_stim>, <start_resp>, <stop_resp>}

Defines the limit line type, the stimulus and response values for a limit line with an arbitrary number of limit line segments.

*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.*

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace. |
| **<type>** | Identifier for the type of the limit line segment. |
| Range [def. unit] | **0** – Limit line segment off, segment defined but no limit check performed<br>**1** – Upper limit line segment<br>**2** – Lower limit line segment |
| **<start_stim>, <stop_stim>, <start_resp>, <stop_resp>** | Stimulus and response values of the first and last points of the limit line segment. |
| Range [def. unit] | Almost no restriction for limit segments; see *Rules for Limit Line Definition.* [Hz for frequency sweeps, dBm for power sweeps, s for time sweeps, see `CALCulate<Chn>:LIMit:CONTrol:DOMain`] |

| | |
|---|---|
| *RST values | – (no limit line defined after *RST) |
| **SCPI, Command Types** | Device-specific, with query. |
| **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 dBm and +3 dBm. |
| | `CALC:LIM:DISP ON` |
| | Show the limit line segment in the active diagram. |

## CALCulate<Chn>:LIMit:DELete:ALL

Deletes all limit line segments.

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace. |
| *RST value | – |
| **SCPI, Command Types** | Device-specific, no query. |
| **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 dBm and +3 dBm. |
| | `CALC:LIM:DEL:ALL` |
| | Delete both created limit line segments. |

## CALCulate<Chn>:LIMit:DISPlay[:STATe] <Boolean>

Displays or hides the entire limit line (including all segments) associated to the active trace.

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace. |
| **<Boolean>** | ON \| OFF - Limit line on or off. |
| *RST value | OFF |
| **SCPI, Command Types** | Device-specific, command or query. |
| **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.

## CALCulate<Chn>:LIMit:FAIL?

Returns a 0 or 1, to indicate whether or not the limit check has failed.

Use `CALCulate<Chn>:CLIMits:FAIL?` to perform a composite (global) limit check.

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace. |
| **Response**<br>*RST value | 0 \| 1 - 0 represents pass, 1 represents fail.<br>0 |
| **SCPI, Command Types** | Confirmed, query only. |
| **Example:** | `*RST; CALC:LIM:CONT 1 GHZ, 2 GHZ`<br>Define an upper limit line segment in the stimulus range between 1 GHz and 2 GHz, using default response values.<br>`CALC:LIM:STAT ON; FAIL?`<br>Switch the limit check on and query the result. |

## CALCulate<Chn>:LIMit:LOWer[:DATA] <numeric_value>,<numeric_value>{,<numeric_value>,<numeric_value>}

Defines the response (y-axis) values of the lower limit line and/or creates new limit line segments.

*The commands `CALCulate<Chn>:LIMit:LOWer[:DATA]` and `CALCulate<Chn>:LIMit:UPPer[:DATA]` use a fixed numbering scheme for limit line segments: Upper limit line segments are assigned odd numbers (1, 3, 5,...), lower limit line segments are assigned even numbers (2, 4, 6,...).*

**Rules for creating segments**

The following rules apply to an active trace with n existing upper and n existing lower 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 lower limit line segments.
- For n > k the response values of all existing lower limit line segments no. 2, 4, ...,2*k are updated, the existing upper and lower limit line segments no. 2*k+1, ..., 2*n are deleted. The existing upper limit line segments no. 1, 3, 2*k–1 are not affected.
- For n < k the response values of the lower limit line segments no. 2, 4 to 2*n are updated, the lower limit line segments 2*n+2, 2*n+4,..., 2*k are generated with default stimulus values (see `CALCulate<Chn>:LIMit:CONTrol[:DATA]`. In addition, the missing upper limit line segments 2*n+1, 2*n+3,..., 2*k–1 are generated with default stimulus and response values

**<Chn>** Channel number used to identify the active trace.

**<numeric_value>** Pair(s) of response values.
Range [def. unit] Almost no restriction for limit segments; see *Rules for Limit Line Definition.* [dB]
*RST value –
The response value of a segment that is created implicitly, e.g. an upper limit line segment, is –20 dB.

**SCPI, Command Types** Confirmed, with query.

**Example:**

```
CALC:LIM:LOW -10, 0, 0, -10
```

Define the following lower and (default) upper limit line segments:

| Seg. | Type | Start Stimulus | Stop Stimulus | Start Response | Stop Response |
|---|---|---|---|---|---|
| 1 | Upper | 300 kHz | 8 GHz | -20 dB | -20 dB |
| 2 | Lower | 300 kHz | 8 GHz | -10 dB | 0 dB |
| 3 | Upper | 300 kHz | 8 GHz | -20 dB | -20 dB |
| 4 | Lower | 300 kHz | 8 GHz | 0 dB | -10 dB |

```
CALC:LIM:DISP ON
```

Show the limit line segments in the active diagram.

## CALCulate<Chn>:LIMit:LOWer:FEED <stimulus_offset>,<response_offset>[,<trace_name>]

Generates a lower limit line using the stimulus values of a data or memory trace and specified offset values.

**<Chn>** Channel number used to identify the active trace. This trace provides the stimulus data for the limit line unless another trace <trace_name> is specified.

**<stimulus_offset>** Stimulus offset value, used to shift all imported limit line segments in horizontal direction.

Range [def. unit] -1000 GHz to +1000 GHz [Hz]
*RST value 0 Hz

**<response_offset>** Response offset value, used to shift all imported limit line segments in vertical direction.
Range [def. unit] $-10^{12}$ dB to $+10^{12}$ dB [dB]
*RST value 0 dB

**<trace_name>** 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>.

**SCPI, Command Types** Device-specific, no query.

**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:LOW:SHIF -3; CALC:LIM:CONT:SHIF 1 GHz
```

Shift the limit line by an additional –3 dB in vertical and by 1 GHz in

horizontal direction. If an upper limit line exists it is also shifted.

## CALCulate<Chn>:LIMit:LOWer:SHIFt <numeric_value>

Shifts all lower and upper limit line segments assigned to the active trace in vertical direction.

*This command is identical with CALCulate<Chn>:LIMit:UPPer:SHIFt.*

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace. |
| **<numeric_value>** | Response offset value for all limit line segments. |
| Range [def. unit] | Almost no restriction for limit segments; see Rules for Limit Line Definition. [dB] |
| *RST value | – |
| **SCPI, Command Types** | Device-specific, no query. |
| **Example:** | `See CALCulate<Chn>:LIMit:LOWer:FEED.` |

## CALCulate<Chn>:LIMit:LOWer:STATe <Boolean>

Switches the lower limit check on or off. Lower limit line segments are assigned even numbers; see `CALCulate<Chn>:LIMit:LOWer[:DATA]`. The command does not affect segments with odd numbers.

Use `CALCulate<Chn>:LIMit:STATe` to switch on or off the entire limit check, including upper and lower limit lines.

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace. |
| **<Boolean>** | ON \| OFF - Limit check on or off. |
| *RST value | OFF |
| **SCPI, Command Types** | Confirmed, command or query. |
| **Example:** | `CALC:LIM:LOW -10, 0, 0, -10` |

Define the following lower and (default) upper limit line segments:

| Seg. | Type | Start Stimulus | Stop Stimulus | Start Response | Stop Response |
|---|---|---|---|---|---|
| 1 | Upper | 300 kHz | 8 GHz | -20 dB | -20 dB |
| 2 | Lower | 300 kHz | 8 GHz | -10 dB | 0 dB |
| 3 | Upper | 300 kHz | 8 GHz | -20 dB | -20 dB |
| 4 | Lower | 300 kHz | 8 GHz | 0 dB | -10 dB |

```
CALC:LIM:LOW:STAT ON; CALC:LIM:FAIL?
```

Switch the limit check on and query the result.

## CALCulate<Chn>:LIMit:RDOMain:COMPlex S | SINV | Y | Z | YREL | ZREL

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`.

This command is complemented by `CALCulate<Chn>:LIMit:RDOMain:FORMat` and `CALCulate<Chn>:LIMit:RDOMain:SPACing`.

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace. |
| **Parameters**<br>Range [def. unit] | Keyword for the physical unit of the response values.<br>The parameters form four groups:<br>▪ S and SINV select relative units (dB) for the limit line.<br>▪ Y selects admittance units (S/Siemens).<br>▪ Z selects impedance units (Ω).<br>▪ YREL and ZREL select dimensionless numbers (U).<br>[–] |
| *RST value | – |
| **SCPI, Command Types** | Device-specific, no query. |

## CALCulate<Chn>:LIMit:RDOMain:FORMat COMPlex | MAGNitude | PHASe | REAL | IMAGinary | SWR | GDELay | L | C

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`.

This command is complemented by `CALCulate<Chn>:LIMit:RDOMain:COMPlex` and `CALCulate<Chn>:LIMit:RDOMain:SPACing`.

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace. |
| **Parameters**<br>Range [def. unit] | Keyword for the physical unit of the response values.<br>The parameters form four groups:<br>▪ COMPlex, REAL, IMAGinary, and SWR select dimensionless numbers (U) for the limit line.<br>▪ MAGNitude selects relative units (dB).<br>▪ PHASe selects phase units (deg).<br>▪ GDELay selects time units (s).<br>▪ L selects inductance units (H/Henry).<br>▪ C selects capacitance units (F/Farad).<br>[–] |
| *RST value | – |
| **SCPI, Command Types** | Device-specific, no query. |

## CALCulate<Chn>:LIMit:RDOMain:SPACing LINear | LOGarithmic | dB | SIC

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`.

This command is complemented by `CALCulate<Chn>:LIMit:RDOMain:COMPlex` and `CALCulate<Chn>:LIMit:RDOMain:FORMat`.

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace. |
| **Parameters** | Keyword for the physical unit of the response values. |
| Range [def. unit] | The analyzer uses dB units, irrespective of the parameter selected. [–] |
| *RST value | – |
| **SCPI, Command Types** | Device-specific, no query. |

## CALCulate<Chn>:LIMit:SEGMent<Seg>:AMPlitude:STARt <numeric_value>

Changes the start response value (i.e. the response value assigned to the start stimulus value) of a limit line segment. A segment must be created first to enable this command (e.g `CALC:LIM:DATA`).

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]`.

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace. |
| **<Seg>** | Segment number |
| **<numeric_value>** | Response value |
| Range [def. unit] | Almost no restriction for limit segments; see *Rules for Limit Line Definition.* [dB] |
| *RST value | – The default response values of a segment that is created by defining its stimulus values only (e.g. by means of `CALCulate<Chn>:LIMit:CONTrol[:DATA]`), is –20 dB. |
| **SCPI, Command Types** | Device-specific, with query. |
| **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 dBm and +3 dBm. |
| | `: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 `2,1000000,2000000,5,5`. |

## CALCulate<Chn>:LIMit:SEGMent<Seg>:AMPlitude:STOP <numeric_value>

Changes the stop response value (i.e. the response value assigned to the stop stimulus value) of a limit line segment. A segment must be created first to enable this command (e.g `CALC:LIM:DATA`).

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]`.

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace. |
| **<Seg>** | Segment number |
| **<numeric_value>** | Response value |
| Range [def. unit] | Almost no restriction for limit segments; see *Rules for Limit Line Definition.* [dB] |
| *RST value | – The default response values of a segment that is created by defining its stimulus values only (e.g. by means of `CALCulate<Chn>:LIMit:CONTrol[:DATA]`), is –20 dB. |
| **SCPI, Command Types** | Device-specific, with query. |
| **Example:** | `See CALCulate<Chn>:LIMit:SEGMent<Seg>:AMPlitude:STARt.` |

## CALCulate<Chn>:LIMit:SEGMent<Seg>:STIMulus:STARt <numeric_value>

Changes the start stimulus value (i.e. the largest or smallest stimulus value) of a limit line segment. A segment must be created first to enable this command (e.g `CALC:LIM:DATA`).

To define the stimulus values of several limit line segments with a single command, use `CALCulate<Chn>:LIMit:CONTrol[:DATA]`.

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace. |
| **<Seg>** | Segment number |
| **<numeric_value>** | Frequency, power or time value, to be defined in accordance with the domain setting (`CALCulate<Chn>:LIMit:CONTrol:DOMain`). |
| Range [def. unit] | Almost no restriction for limit segments; see *Rules for Limit Line Definition.* In particular, the start value can be larger than the stop value `CALCulate<Chn>:LIMit:SEGMent<Seg>:STIMulus:STOP`. [Hz for frequency sweeps, dBm for power sweeps, s for time sweeps] |
| *RST value | – A segment that is created implicitly, e.g. by means of `CALCulate<Chn>:LIMit:UPPer[:DATA]` or `CALCulate<Chn>:LIMit:LOWer[:DATA,]`, covers the maximum sweep range of the analyzer. |
| **SCPI, Command Types** | Device-specific, with query. |
| **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 dBm and +3 |

dBm.

```
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 `2,1000000,2000000,2,3`.

## CALCulate<Chn>:LIMit:SEGMent<Seg>:STIMulus:STOP <numeric_value>

Changes the stop stimulus value (i.e. the largest or smallest stimulus value) of a limit line segment. A segment must be created first to enable this command (e.g `CALC:LIM:DATA`).

To define the stimulus values of several limit line segments with a single command, use `CALCulate<Chn>:LIMit:CONTrol[:DATA]`.

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace. |
| **<Seg>** | Segment number |
| **<numeric_value>** | Frequency, power or time value, to be defined in accordance with the domain setting (`CALCulate<Chn>:LIMit:CONTrol:DOMain`). |
| Range [def. unit] | Almost no restriction for limit segments; see *Rules for Limit Line Definition.* In particular, the stop value can be smaller than the start value `CALCulate<Chn>:LIMit:SEGMent<Seg>:STIMulus:START`.<br>[Hz for frequency sweeps, dBm for power sweeps, s for time sweeps] |
| *RST value | –<br>A segment that is created implicitly, e.g. by means of `CALCulate<Chn>:LIMit:UPPer[:DATA]` or `CALCulate<Chn>:LIMit:LOWer[:DATA,]`, covers the maximum sweep range of the analyzer. |
| **SCPI, Command Types** | Device-specific, with query. |
| **Example:** | `See CALCulate<Chn>:LIMit:SEGMent<Seg>:STIMulus:STARt.` |

## CALCulate<Chn>:LIMit:SEGMent<Seg>:TYPE LMIN | LMAX | OFF

Selects the limit line type for a limit line segment. This 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`).

*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 LIMit subsystem defining stimulus and response values of limit lines.*

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace. |
| **<Seg>** | Segment number |
| **Parameters** | Limit line type |

| | |
|---|---|
| Range [def. unit] | LMAX (upper limit line segment), LMIN (lower limit line segment), OFF (limit check switched off, limit line segment not deleted) [–] |
| *RST value | LMAX |
| **SCPI, Command Types** | Device-specific, with query. |
| **Example:** | `*RST; CALC:LIM:UPP 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:SEG:TYPE LMIN` |
| | Turn the defined limit line segment into a lower limit line segment. |

## CALCulate<Chn>:LIMit:SOUNd[:STATe]

Switches the acoustic signal (fail beep) on or off. The fail beep is generated each time the analyzer detects an exceeded limit.

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace. |
| **<Boolean>** | ON \| OFF - Fail beep on or off. |
| *RST value | OFF |
| **SCPI, Command Types** | Confirmed, command or query. |
| **Example:** | `CALC:LIM:STAT ON; SOUN ON` |
| | Switch the limit check on and activate the fail beep. |

## CALCulate<Chn>:LIMit:STATe

Switches the limit check (including upper and lower limits) on or off.

Use `CALCulate<Chn>:LIMit:UPPer:STATe` or `CALCulate<Chn>:LIMit:LOWer:STATe` to switch on or off the individual limit checks for upper or lower limit lines.

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace. |
| **<Boolean>** | ON \| OFF - Limit check on or off. |
| *RST value | OFF |
| **SCPI, Command Types** | Confirmed, command or query. |
| **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; CALC:LIM:FAIL?` |
| | Switch the limit check on and query the result. |

## CALCulate<Chn>:LIMit:TTLout<Output_no>[:STATe] <Boolean>

Switches the TTL pass/fail signals on or off. The signals are applied to the USER CONTROL connector as long as the active trace <Chn> is within limits.

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace. |
| **<Output_no>** | 1 – TTL out pass 1 (pin 13 of USER CONTROL connector)<br>2 – TTL out pass 2 (pin 14 of USER CONTROL connector) |
| **<Boolean>**<br>*RST value | ON \| OFF - TTL output signal on or off.<br>OFF |
| **SCPI, Command Types** | Device-specific, command or query. |
| **Example:** | `*RST; CALC:LIM:CONT 1 GHZ, 2 GHZ`<br>Define an upper limit line segment in the stimulus range between 1 GHz and 2 GHz, using default response values.<br>`CALC:LIM:STAT ON; TTL2 ON`<br>Switch the limit check on and activate the TTL out pass 2 signal. |

## CALCulate<Chn>:LIMit:UPPer[:DATA] <numeric_value>,<numeric_value>{,<numeric_value>,<numeric_value>}

Defines the response (y-axis) values of the upper limit line and/or creates new limit line segments.

*The commands `CALCulate<Chn>:LIMit:LOWer[:DATA]` and `CALCulate<Chn>:LIMit:UPPer[:DATA]` use a fixed numbering scheme for limit line segments: Upper limit line segments are assigned odd numbers (1, 3, 5,...), lower limit line segments are assigned even numbers (2, 4, 6,...).*

**Rules for creating segments**

The following rules apply to an active trace with n existing upper and n existing lower 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 upper limit line segments.
- For n > k the response values of all existing upper limit line segments no. 1, 3, ...,2*k–1 are updated, the existing upper and lower limit line segments no. 2*k+1, ..., 2*n are deleted. The existing lower limit line segments no. 2, 4, 2*k are not affected.
- For n < k the response values of the upper limit line segments no. 1, 3 to 2*n–1 are updated, the upper limit line segments 2*n+1, 2*n+3,..., 2*k–1 are generated with default stimulus values (see `CALCulate<Chn>:LIMit:CONTrol[:DATA]`. In addition, the missing lower limit line segments 2*n+2, 2*n+4,..., 2*k are generated with default stimulus and response values

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace. |
| **<numeric_value>** | Pair(s) of response values. |
| Range [def. unit] | Almost no restriction for limit segments; see *Rules for Limit Line Definition.*<br>[dB] |

*RST value –
The response value of a segment that is created implicitly, e.g. an lower limit line segment, is –20 dB.

**SCPI, Command Types** Confirmed, with query.

**Example:**

```
CALC:LIM:UPP -10, 0, 0, -10
```

Define the following upper and (default) lower limit line segments:

| Seg. | Type | Start Stimulus | Stop Stimulus | Start Response | Stop Response |
|---|---|---|---|---|---|
| 1 | Upper | 300 kHz | 8 GHz | -10 dB | 0 dB |
| 2 | Lower | 300 kHz | 8 GHz | -20 dB | -20 dB |
| 3 | Upper | 300 kHz | 8 GHz | 0 dB | -10 dB |
| 4 | Lower | 300 kHz | 8 GHz | -20 dB | -20 dB |

```
CALC:LIM:DISP ON
```

Show the limit line segments in the active diagram.

## CALCulate<Chn>:LIMit:UPPer:FEED

Generates an upper limit line using the stimulus values of a data or memory trace and specified offset values.

**<Chn>** Channel number used to identify the active trace. This trace provides the stimulus data for the limit line unless another trace <trace_name> is specified.

**<stimulus_offset>** Stimulus offset value, used to shift all imported limit line segments in horizontal direction.

Range [def. unit] -1000 GHz to +1000 GHz [Hz]
*RST value 0 Hz

**<response_offset>** Response offset value, used to shift all imported limit line segments in vertical direction.

Range [def. unit] $-10^{12}$ dB to $+10^{12}$ dB [dB]
*RST value 0 dB

**<trace_name>** 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>.

**SCPI, Command Types** Device-specific, no query.

**Example:**

```
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:UPP:SHIF 3; CALC:LIM:CONT:SHIF 1 GHz
```

Shift the limit line by an additional 3 dB in vertical and by 1 GHz in horizontal direction. If a lower limit line exists it is also shifted.

## CALCulate<Chn>:LIMit:UPPer:SHIFt <numeric_value>

Shifts all lower and upper limit line segments assigned to the active trace in vertical direction.

NOTE This command is identical with `CALCulate<Chn>:LIMit:LOWer:SHIFt`.

**<Chn>** Channel number used to identify the active trace.

**<numeric_value>** Response offset value for all limit line segments.

Range [def. unit] Almost no restriction for limit segments; see Rules for Limit Line Definition. [dB]

*RST value –

**SCPI, Command Types** Device-specific, no query.

**Example:**

```
See CALCulate<Chn>:LIMit:UPPer:FEED.
```

## CALCulate<Chn>:LIMit:UPPer:STATe <Boolean>

Switches the upper limit check on or off. Upper limit line segments are assigned odd numbers; see `CALCulate<Chn>:LIMit:UPPer[:DATA]`. The command does not affect segments with even numbers.

TIP Use `CALCulate<Chn>:LIMit:STATe` to switch on or off the entire limit check, including upper and lower limit lines.

**<Chn>** Channel number used to identify the active trace.

**<Boolean>** ON | OFF - Limit check on or off.

*RST value OFF

**SCPI, Command Types** Confirmed, command or query.

**Example:**

```
CALC:LIM:UPP -10, 0, 0, -10
```

Define the following upper and (default) lower limit line segments:

| Seg. | Type | Start Stimulus | Stop Stimulus | Start Response | Stop Response |
|---|---|---|---|---|---|
| 1 | Upper | 300 kHz | 8 GHz | -10 dB | 0 dB |
| 2 | Lower | 300 kHz | 8 GHz | -20 dB | -20 dB |
| 3 | Upper | 300 kHz | 8 GHz | 0 dB | -10 dB |
| 4 | Lower | 300 kHz | 8 GHz | -20 dB | -20 dB |

```
CALC:LIM:UPP:STAT ON; CALC:LIM:FAIL?
```

Switch the limit check on and query the result.

## CALCulate<Chn>:MARKer<Mk>...

This subsystem controls the marker functions. The commands are device-specific and beyond what is specified in the SCPI subsystem SOURce:MARKer.

## CALCulate<Chn>:MARKer<Mk>:AOFF

Removes all markers from all traces of the active setup. The removed markers remember their properties (stimulus value, format, delta mode, number) when they are restored (CALC<Chn>:MARK<Mk> ON). The marker properties are definitely lost if the associated trace is deleted.

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace. If unspecified the numeric suffix is set to 1. |
| **<Mk>** | Marker number in the range 1 to 10. This numeric suffix is ignored and may be set to any value. |
| *RST value | – |
| **SCPI, Command Types** | Device-specific, no query. |
| **Example:** | `Suppose that the active setup contains an active trace no. 1.`<br>`CALC:MARK1 ON; MARK2 ON`<br>Create markers 1 and 2 and assign them to the trace no. 1.<br>`CALC:MARK:AOFF`<br>Remove both markers. |

## CALCulate<Chn>:MARKer<Mk>:BWIDth <x_dB_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).

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace. |
| **<Mk>** | Marker number in the range 1 to 10. This numeric suffix is ignored and may be set to any value because the bandfilter search functions always use markers Mkr1 to Mkr4. |
| **<x_dB_Bandwidth>** | *x dB Bandwidth* parameter. The *x dB Bandwidth* is the difference between the band edges and the center response value of a bandfilter peak; it must be negative for a bandpass search and positive for a bandstop search. |
| Range [def. unit] | For bandpass: –100.00 dB to –0.01 dB, the increment (UP/DOWN) is 0.3 dB. [dB]<br>For bandstop: +0.01 dB to +100.00 dB, the increment (UP/DOWN) is 0.3 dB. [dB] |
| *RST value | –3 dB |
| **Response for** | Search result in the format <Bandwidth>, <Center_Stimulus>, <Q>, |

| | |
|---|---|
| **query:** | <Loss>, <LBE>, <UBE> where: |
| **<Bandwidth>** | n-dB bandwidth of the bandpass/bandstop region, where n is the selected *x dB Bandwidth* (setting command `CALCulate<Chn>:MARKer<Mk>:BWIDth <x dB Bandwidth>`). |
| **<Center_Stimulus>** | Stimulus frequency at the center of the bandpass/bandstop region (the stimulus value of marker no. 4). |
| **<Q>** | 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 no. 4 at the time of the bandfilter search). |
| **<LBE>** | Lower band edge |
| **<UBE>** | Upper band edge |
| Range [def. unit] | Depending on the measurement parameter. [units depending on the sweep type and the marker format; see `CALCulate<Chn>:MARKer<Mk>:FORMat`] |
| *RST value | – |
| **SCPI, Command Types** | Device-specific, query only. |
| **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 1 to 4. |
| | `CALC:MARK:SEAR:BFIL:RES ON` |
| | Display the marker info field in the diaram 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. |

## CALCulate<Chn>:MARKer<Mk>:COUPled[:STATe] <Boolean>

Couples the markers of all traces in the active setup to the markers of trace no. <Chn>, provided that they have the same sweep type (`SENSe<Chn>:FUNCtion`).

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace. The effects of marker coupling depend on the active trace number; see *Coupled Markers* in the GUI Reference chapter. |
| **<Mk>** | Marker number in the range 1 to 10. This suffix is ignored because the command affects all markers. |
| **<Boolean>** | ON \| OFF - Enables or disables marker coupling. |
| *RST value | OFF |
| **SCPI, Command Types** | Device-specific, command or query. |
| **Example:** | |

Suppose that the active setup contains two traces Trc1 and Trc2, assigned to channels no. 1 and 2, respectively.

```
:CALC1:PAR:SEL 'TRC1'; CALC1:MARK1 ON; MARK2 ON
```

Select Trc1 as the active trace and create the two markers no. 1 and 2. The default position for both markers is the center of the sweep range.

```
CALC1:MARK:COUP ON
```

Create two markers no. 1 and 2 on Trc 2 and couple them to the markers of Trc 1.

## 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.

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace. |
| **<Mk>** | Marker number in the range 1 to 10. |
| **<Boolean>**<br>*RST value | ON \| OFF - Enables or disables the delta mode.<br>OFF |
| **SCPI, Command Types** | Device-specific, command or query. |
| **Example:** | |

Suppose that the active setup 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.

## CALCulate<Chn>:MARKer<Mk>:FORMat MLINear | MLOGarithmic | PHASe | POLar | COMPlex | GDELay | REAL | IMAGinary | SWR | LINPhase | LOGPhase | IMPedance | ADMittance | MDB | MLPHase | MDPHase

Defines the output format for the (complex) value of marker <Mk> on trace no. <Chn>.

***Note:*** *The formats of the markers assigned to a trace are independent of each other and of the trace format settings; see `CALCulate<Chn>:FORMat`.*

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace. |
| **<Mk>** | Marker number in the range 1 to 10. |
| **Parameters** | See list of parameters below. |
| *RST value | DEF – This means the format of the trace no. <Chn> (MLOG after *RST); see `CALCulate<Chn>:FORMat`. |
| **SCPI, Command Types** | Device-specific, command or query |
| **Example:** | |

Suppose that the active setup contains an active trace no. 1.

```
CALC:MARK ON; CALC:MARK:FORM?
```

Create marker 1, assign it to the trace no. 1 and query its format. The analyzer returns the format of the active trace.

Assume that the marker result is given by the complex quantity z = x + jy. The meaning of the parameters is as follows:

| | |
|---|---|
| MLINear | \|z\| = sqrt ( x2 + y2 ) |
| MLOGarithmic<br>ZVR MDB | \|z\| = sqrt ( x2 + y2 )<br>dB Mag(z) = 20 * log\|z\| dB |
| PHASe | ϕ (z) = arctan ( Im(z) / Re(z) ) |
| POLar<br>ZVR COMPlex | x, y (Real and Imag) |
| GDELay | Group Delay, – d ϕ (z)/dω |
| REAL | x |
| IMAGinary | y |
| SWR | Standing Wave Ratio, SWR = (1 + \|z\|) / (1 – \|z\|) |
| LINPhase<br>ZVR MLPhase | Lin Mag and Phase, \|z\|, arctan ( Im(z) / Re(z) ) |
| LOGPhase<br>ZVR MDPhase | dB Mag and Phase, 20 * log\|z\| dB, arctan ( Im(z) / Re(z) ) |
| IMPedance | R, X, L or C (depending on sign(X)) |
| ADMittance | G, B, L/C (depending on sign(X)) |

## CALCulate<Chn>:MARKer<Mk>:FUNCtion:BWIDth <x dB Bandwidth>

Defines the bandfilter level, i.e. the minimum excursion for the bandpass and bandstop peaks.

Use `CALCulate<Chn>:MARKer<Mk>:BWIDth` to set the *x dB Bandwidth* and query the results of a bandfilter search. Note the sign convention for input values.

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace. |
| **<Mk>** | Marker number in the range 1 to 10. This numeric suffix is ignored and may be set to any value because the bandfilter search functions always use markers Mkr1 to Mkr4. |
| **<x dB Bandwidth>** | *x dB Bandwidth* parameter. |
| Range [def. unit] | 0.01 dB to 100.00 dB, the increment (UP. DOWN) is 0.3 dB. [dB] |
| *RST value | 3 dB. |
| **SCPI, Command Types** | Device-specific, command or query. |
| **Example:** | `See CALCulate<Chn>:MARKer<Mk>:BWIDth.` |

## CALCulate<Chn>:MARKer<Mk>:FUNCtion:BWIDth:MODE BPASs | BSTop | BPRMarker | BSRMarker

Selects the bandfilter search mode.

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace. |
| **<Mk>** | Marker number in the range 1 to 10. This numeric suffix is ignored and may be set to any value because the bandfilter search functions always use markers Mkr1 to Mkr4. |
| **Parameters** | Bandfilter search type:<br>BPASs: Bandpass Search Ref to Max<br>BSTop: Bandstop Search Ref to Max<br>BPRMarker: Bandpass Search Ref to Marker<br>BSRMarker: Bandstop Search Ref to Marker |
| *RST value | BPASs |
| **SCPI, Command Types** | Device-specific, command or query. |
| **Example:** | `See CALCulate<Chn>:MARKer<Mk>:BWIDth.` |

## CALCulate<Chn>:MARKer<Mk>:FUNCtion:CENTer

Sets the center of the sweep range equal to the stimulus value of the marker <Mk> on trace no. <Chn>.

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace. |
| **<Mk>** | Marker number in the range 1 to 10. |
| Range [def. unit] | – [–] |
| *RST value | – |
| **SCPI, Command Types** | Device-specific, no query. |
| **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. |

## 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].`*

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace. |
| **<Mk>** | Marker number in the range 1 to 10. |
| **<Boolean>** | ON \| OFF - Enables or disables the delta mode. |
| *RST value | OFF |

**SCPI, Command Types** Device-specific, command or query.

## CALCulate<Chn>:MARKer<Mk>:FUNCtion:DOMain:USER <numeric_value>

Assigns a search range no. <numeric_value> to marker no <Mk> and selects the search range, e.g. in order to define the start and stop values.

**<Chn>** Channel number used to identify the active trace.

**<Mk>** Marker number in the range 1 to 10.

**<numeric_value>** Number of the search range.
Range [def. unit] 0 – fixed full span search range (equal to the sweep range)
1 to 10 – user-definable search ranges; see example
*RST value 0 (reserved for full span search range)

**SCPI, Command Types** Device-specific, command or query.

**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.

## CALCulate<Chn>:MARKer<Mk>:FUNCtion:DOMain:USER:STARt <numeric_value>

Defines the start value of the search range selected with `CALCulate<Chn>:MARKer<Mk>:FUNCtion:DOMain:USER <numeric_value>`.

**<Chn>** Channel number used to identify the active trace.

**<Mk>** Marker number in the range 1 to 10.

**<numeric_value>** Beginning of the search range.
Range [def. unit] Maximum allowed sweep range, depending on the instrument model and on the sweep type. [Hz, dBm or s, depending on the sweep type]
*RST value 0 Hz

**SCPI, Command Types** Device-specific, command or query.

**Example:**

```
See CALCulate<Chn>:MARKer<Mk>:FUNCtion:DOMain:USER <numeric_value>
```

## CALCulate<Chn>:MARKer<Mk>:FUNCtion:DOMain:USER:STOP <numeric_value>

Defines the stop value of the search range selected with `CALCulate<Chn>:MARKer<Mk>:FUNCtion:DOMain:USER <numeric_value>`.

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace. |
| **<Mk>** | Marker number in the range 1 to 10. |
| **<numeric_value>** | End of the search range. |
| Range [def. unit] | Maximum allowed sweep range, depending on the instrument model and on the sweep type. [Hz, dBm or s, depending on the sweep type] |
| *RST value | 0 Hz |
| **SCPI, Command Types** | Device-specific, command or query. |
| **Example:** | `See CALCulate<Chn>:MARKer<Mk>:FUNCtion:DOMain:USER <numeric_value>` |

## CALCulate<Chn>:MARKer<Mk>:FUNCtion:EXECute MAXimum | MINimum | RPEak | LPEak | NPEak | TARGet | LTARget | RTARget | BFILter

Selects a search mode for marker no. <Mk> and initiates the search. The marker must be created before using `CALCulate<Chn>:MARKer<Mk>[:STATe] ON` (exception: bandfilter search).

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace. |
| **<Mk>** | Marker number in the range 1 to 10. For a bandfilter search (`BFILter`) this numeric suffix is ignored and may be set to any value because the bandfilter search functions always use markers Mkr1 to Mkr4. |
| **Parameters** | See list of parameters below. |
| *RST value | – |
| **SCPI, Command Types** | Device-specific, no query. |
| **Example:** | |

Suppose that the active setup contains an active trace no. 1.

```
CALC:MARK ON
```

Create marker 1 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.

The analyzer provides the following search modes:

| Mode | Find... |
|---|---|
| `MAXimum` | Absolute maximum in the search range (see `CALCulate<Chn>:MARKer<Mk>:FUNCtion:DOMain:USER <numeric_value>`) |
| `MINimum` | Absolute maximum in the search range |
| `RPEak` | Next valid peak to the right of the current marker position |
| `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` | Bandfilter search. The results are queried using `CALCulate<Chn>:MARKer<Mk>:BWIDth`. |

## CALCulate<Chn>:MARKer<Mk>:FUNCtion:RESult?

Returns the result (stimulus and response value) of a search started by means of `CALCulate<Chn>:MARKer<Mk>:FUNCtion:EXECute`. The search must be executed before the command is enabled.

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace. |
| **<Mk>** | Marker number in the range 1 to 10. This numeric suffix is ignored and may be set to any value. |
| **Response** | Search result in the format <Stimulus value>, <Response_value>. |
| Range [def. unit] | Depending on the measurement parameter. [unit depending on the sweep type and the marker format; see `CALCulate<Chn>:MARKer<Mk>:FORMat`] |
| *RST value | – |
| **SCPI, Command Types** | Device-specific, query only. |
| **Example:** | `See CALCulate<Chn>:MARKer<Mk>:FUNCtion:EXECute.` |

## CALCulate<Chn>:MARKer<Mk>:FUNCtion[:SELect] MAXimum | MINimum | RPEak | LPEak | NPEak | TARGet | LTARget | RTARget | BFILter

Selects a search mode for marker no. <Mk>, which can then be initiated using one of the `CALCulate<Chn>:MARKer<Mk>:SEARch...`, `CALCulate<Chn>:MARKer<Mk>:MAXimum` or `CALCulate<Chn>:MARKer<Mk>:MINimum` functions. The marker must be created before using `CALCulate<Chn>:MARKer<Mk>[:STATe] ON`.

*Note: This command is not needed expect for compatibility with ZVR programs. Use `CALCulate<Chn>:MARKer<Mk>:FUNCtion:EXECute` to select a search mode and at the same time initiate the search. The `CALCulate<Chn>:MARKer<Mk>:SEARch...`, `CALCulate<Chn>:MARKer<Mk>:MAXimum` or `CALCulate<Chn>:MARKer<Mk>:MINimum` functions also select the search mode.*

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace. |
| **<Mk>** | Marker number in the range 1 to 10. For a bandfilter search (`BFILter`) this numeric suffix is ignored and may be set to any value because the bandfilter search functions always use markers Mkr1 to Mkr4. |
| **Parameters** | See list of parameters below. |
| *RST value | – |
| **SCPI, Command Types** | Device-specific, command or query. |

The analyzer provides the following search modes:

| Mode | Selected search mode |
|---|---|
| `MAXimum` | Absolute maximum in the search range (see `CALCulate<Chn>:MARKer<Mk>:FUNCtion:DOMain:USER <numeric_value>`) |
| `MINimum` | Absolute maximum in the search range |
| `RPEak` | Next valid peak to the right of the current marker position |
| `LPEak` | Next valid peak to the left |
| `NPEak` | Next highest value among the valid peaks |
| `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 | Bandfilter search. Bandfilter search. The results are queried using CALCulate<Chn>:MARKer<Mk>:BWIDth. |

## CALCulate<Chn>:MARKer<Mk>:FUNCtion:STARt

Sets the beginning (start) of the sweep range equal to the stimulus value of the marker <Mk> on trace no. <Chn>.

**<Chn>** Channel number used to identify the active trace.

**<Mk>** Marker number in the range 1 to 10.

Range [def. unit] – [–]
*RST value –

**SCPI, Command Types** Device-specific, no query.

**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.

## CALCulate<Chn>:MARKer<Mk>:FUNCtion:STOP

Sets the end (stop) of the sweep range equal to the stimulus value of the marker <Mk> on trace no. <Chn>.

**<Chn>** Channel number used to identify the active trace.

**<Mk>** Marker number in the range 1 to 10.

Range [def. unit] – [–]
*RST value –

**SCPI, Command Types** Device-specific, no query.

**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:STOP
```

Divide the sweep range in half, ending at the current marker position.

## CALCulate<Chn>:MARKer<Mk>:FUNCtion:TARget <numeric_value>

Defines the target value for the target search of marker no. <Mk>, which can be activated using CALCulate<Chn>:MARKer<Mk>:FUCTion:EXECute TARGet.

***Note:** This command is the ZVR-compatible equivalent of CALCulate<Chn>:MARKer<Mk>:TARget.*

**<Chn>** Channel number used to identify the active trace.

**<Mk>** Marker number in the range 1 to 10.

| | |
|---|---|
| **<numeric_value>** | Target search value of marker no. <Mk>. |
| Range [def. unit] | Depending on the format of the active trace (`CALCulate<Chn>:FORMat`). For a dB Mag trace the range is –300 dB to +200 dB, the increment (UP. DOWN) is 0.1 dB. [dB, deg etc., depending on the trace format] |
| *RST value | Depending on the trace format; 0 dB for a dB Mag trace. |
| **SCPI, Command Types** | Device-specific, command or query. |

## CALCulate<Chn>:MARKer<Mk>:MAXimum

Selects a search mode for marker no. <Mk> and initiates a maximum search. The marker must be created before using `CALCulate<Chn>:MARKer<Mk>[:STATe] ON`.

***Note:*** *This command is the ZVR-compatible equivalent of*
*`CALCulate<Chn>:MARKer<Mk>:FUNCtion:EXECute MAXimum`.*

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace. |
| **<Mk>** | Marker number in the range 1 to 10. |
| Range [def. unit] | – [–] |
| *RST value | – |
| **SCPI, Command Types** | Device-specific, no query. |

## CALCulate<Chn>:MARKer<Mk>:MINimum

Selects a search mode for marker no. <Mk> and initiates a minimum search. The marker must be created before using `CALCulate<Chn>:MARKer<Mk>[:STATe] ON`.

***Note:*** *This command is the ZVR-compatible equivalent of*
*`CALCulate<Chn>:MARKer<Mk>:FUNCtion:EXECute MAXimum`.*

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace. |
| **<Mk>** | Marker number in the range 1 to 10. |
| Range [def. unit] | – [–] |
| *RST value | – |
| **SCPI, Command Types** | Device-specific, no query. |

## CALCulate<Chn>:MARKer<Mk>:MODE CONTinuous | DISCrete

Sets marker no. <Mk> to continuous or discrete mode. The marker doesn't have to be created before (`CALCulate<Chn>:MARKer<Mk>[:STATe] ON`), the mode can be assigned in advance.

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace. |
| **<Mk>** | Marker number in the range 1 to 10. |
| **Parameters** | CONTinuous – marker can be positioned on any point of the trace, and its response values are obtained by interpolation.<br>DISCrete – marker can be set to discrete sweep points only. |
| *RST value | CONT |
| **SCPI, Command** | Device-specific, command or query. |

**Types**
**Example:**

Suppose that the active setup contains an active trace no. 1.

```
CALC:MARK:MODE DISC; MARK2 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 position can be different.

## CALCulate<Chn>:MARKer<Mk>:REFerence[:STATe] <Boolean>

Creates the reference marker and assigns it to trace no. <Chn>.

**<Chn>** Channel number used to identify the active trace.

**<Mk>** Marker number in the range 1 to 10. This numeric suffix is ignored and may be set to any value.

**<Boolean>** ON | OFF - Creates or removes the marker.
*RST value OFF

**SCPI, Command Types**
**Example:**

Device-specific, command or query.

Suppose that the active setup contains an active trace no. 1.

```
CALC:MARK:REF ON; CALC:MARK ON
```

Create the reference marker and marker 1 and assign them to trace no. 1. The default position of both markers is the center of the sweep range.

## CALCulate<Chn>:MARKer<Mk>:REFerence:X <numeric_value>

Defines the stimulus (in Cartesian diagrams: x-axis) value of the reference marker, which can (but doesn't have to) be displayed using `CALCulate<Chn>:MARKer<Mk>:REFerence[:STATe] ON`.

**<Chn>** Channel number used to identify the active trace.

**<Mk>** Marker number in the range 1 to 10. This numeric suffix is ignored and may be set to any value.

**<numeric_value>** Stimulus value of the reference marker.
Range [def. unit] –9.9E+11 Hz to +9.9E+11 Hz [Hz] (for frequency sweeps)
-999 dBm to +999 dBm [dBm] (for power sweeps)
0 s to 127500 s [s] (for time and CW mode sweeps)
*RST value 0 Hz

**SCPI, Command Types**
**Example:**

Device-specific, command or query.

Suppose that the active setup contains an active trace no. 1 and that the sweep range for a frequency sweep starts at 1 GHz.

```
CALC:MARK:REF ON
```

Create the reference marker and display it in the center of the sweep

range.

```
CALC:MARK:REF:X 1GHz
```

Set the reference marker to the beginning of the sweep range.

## CALCulate<Chn>:MARKer<Mk>:REFerence:Y?

Returns the response (in Cartesian diagrams: y-axis) value of the reference marker. The reference marker must be created before using `CALCulate<Chn>:MARKer<Mk>:REFerence[:STATe] ON`.

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace. |
| **<Mk>** | Marker number in the range 1 to 10. This numeric suffix is ignored and may be set to any value. |
| **Response** | Response value of the reference marker. |
| Range [def. unit] | Depending on the measured quantity. [unit depending on the marker format; see `CALCulate<Chn>:MARKer<Mk>:FORMat`] |
| *RST value | – |
| **SCPI, Command Types** | Device-specific, query only. |
| **Example:** | Suppose that the active setup contains an active trace no. 1. |

```
CALC:MARK:REF ON
```

Create the reference marker and display it in the center of the sweep range.

```
CALC:MARK:REF:Y?
```

Query the measurement value at the reference marker position.

## CALCulate<Chn>:MARKer<Mk>SEARch:BFILter:RESult[:STATe] <Boolean>

Shows or hides the bandfilter search results in the diagram area.

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace. |
| **<Mk>** | Marker number in the range 1 to 10. This numeric suffix is ignored and may be set to any value. |
| **<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.<br>OFF - hide the bandfilter search results. |
| *RST value | OFF |
| **SCPI, Command Types** | Device-specific, command or query. |
| **Example:** | |

```
See CALCulate<Chn>:MARKer<Mk>:BWIDth.
```

## CALCulate<Chn>:MARKer<Mk>:SEARch[:IMMediate]

Initiates a search according to the search function selected with `CALCulate<Chn>:MARKer<Mk>:FUNCtion[:SELect]`. The marker must be created before using `CALCulate<Chn>:MARKer<Mk>[:STATe] ON`.

***Note:*** *Together with `CALCulate<Chn>:MARKer<Mk>:FUNCtion[:SELect]` this command is the ZVR-compatible equivalent of `CALCulate<Chn>:MARKer<Mk>:FUNCtion:EXECute ...`.*

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace. |
| **<Mk>** | Marker number in the range 1 to 10. For a bandfilter search (`BFILter`) this numeric suffix is ignored and may be set to any value because the bandfilter search functions always use markers Mkr1 to Mkr4. |
| Range [def. unit] | – [–] |
| *RST value | – |
| **SCPI, Command Types** | Device-specific, no query. |

## CALCulate<Chn>:MARKer<Mk>:SEARch:LEFT

Selects a search mode for marker no. <Mk> and initiates a search for the next valid peak to the left. The marker must be created before using `CALCulate<Chn>:MARKer<Mk>[:STATe] ON`.

***Note:*** *This command is the ZVR-compatible equivalent of `CALCulate<Chn>:MARKer<Mk>:FUNCtion:EXECute LPEak`.*

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace. |
| **<Mk>** | Marker number in the range 1 to 10. |
| Range [def. unit] | – [–] |
| *RST value | – |
| **SCPI, Command Types** | Device-specific, no query. |

## CALCulate<Chn>:MARKer<Mk>:SEARch:NEXT

Selects a search mode for marker no. <Mk> and initiates a search for the next highest or lowest value among the valid peaks. The marker must be created before using `CALCulate<Chn>:MARKer<Mk>[:STATe] ON`.

***Note:*** *This command is the ZVR-compatible equivalent of `CALCulate<Chn>:MARKer<Mk>:FUNCtion:EXECute NPEak`.*

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace. |
| **<Mk>** | Marker number in the range 1 to 10. |
| Range [def. unit] | – [–] |
| *RST value | – |
| **SCPI, Command Types** | Device-specific, no query. |

## CALCulate<Chn>:MARKer<Mk>:SEARch:RIGHt

Selects a search mode for marker no. <Mk> and initiates a search for the next valid peak to the right. The marker must be created before using `CALCulate<Chn>:MARKer<Mk>[:STATe] ON`.

***Note:*** *This command is the ZVR-compatible equivalent of*
*`CALCulate<Chn>:MARKer<Mk>:FUNCtion:EXECute RPEak`.*

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace. |
| **<Mk>** | Marker number in the range 1 to 10. |
| Range [def. unit] | – [–] |
| *RST value | – |
| **SCPI, Command Types** | Device-specific, no query. |

## CALCulate<Chn>:MARKer<Mk>:SEARch:TRACking <Boolean>

Enables or disables the marker tracking mode for marker no. <Mk>. Tracking mode causes the active minimum/maximum or target search of the active marker to be repeated after each sweep. A marker must be created and a search mode must be active (`CALCulate<Chn>:MARKer<Mk>:FUNCtion:EXECute ...`) to use this command.

If the current search mode is a bandfilter search this command enables or disables bandfilter tracking.

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace. |
| **<Mk>** | Marker number in the range 1 to 10. For a bandfilter search (`BFILter`) this numeric suffix is ignored and may be set to any value because the bandfilter search functions always use markers Mkr1 to Mkr4. |
| **<Boolean>** | ON \| OFF - Enables or disables the marker tracking mode. |
| *RST value | OFF |
| **SCPI, Command Types** | Device-specific, command or query. |
| **Example:** | |
| | Suppose that the active setup contains an active trace no. 1. |
| | `CALC:MARK ON; FUNC:EXEC MAXimum` |
| | Create marker no. 1 and assign them 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. |

## CALCulate<Chn>:MARKer<Mk>[:STATe] <Boolean>

Creates the marker numbered <Mk> and assigns it to trace no. <Chn>.

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace. |
| **<Mk>** | Marker number in the range 1 to 10. If unspecified the numeric suffix is set to |

1.

**<Boolean>** ON | OFF - Creates or removes the marker.
*RST value OFF

**SCPI, Command Types** Device-specific, command or query.

**Example:**

```
Suppose that the active setup 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.
```

## CALCulate<Chn>:MARKer<Mk>:TARget <numeric_value>

Defines the target value for the target search of marker no. <Mk>, which can be activated using `CALCulate<Chn>:MARKer<Mk>:FUCTion:EXECute TARGet`.

**<Chn>** Channel number used to identify the active trace.

**<Mk>** Marker number in the range 1 to 10.

**<numeric_value>** Target search value of marker no. <Mk>.
Range [def. unit] Depending on the format of the active trace (`CALCulate<Chn>:FORMat`). For a dB Mag trace the range is –300 dB to +200 dB, the increment (UP. DOWN) is 0.1 dB. [dB, deg etc., depending on the trace format]
*RST value Depending on the trace format; 0 dB for a dB Mag trace.

**SCPI, Command Types** Device-specific, command or query.

**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.

## CALCulate<Chn>:MARKer<Mk>:TYPE NORMal | FIXed

Sets marker no. <Mk> to normal or fixed mode. The marker must be created before using `CALCulate<Chn>:MARKer<Mk>[:STATe] ON`.

**<Chn>** Channel number used to identify the active trace.

**<Mk>** Marker number in the range 1 to 10.

**Parameters** NORMal – response value changes according to the measurement result.
FIXed – marker keeps its current response value.
*RST value NORMal

**SCPI, Command Types** Device-specific, command or query.

**Example:**

```
CALC:MARK ON; CALC:MARK:TYPE FIX
```

Create marker 1and 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.

## CALCulate<Chn>:MARKer<Mk>:X <numeric_value>

Defines the stimulus (in Cartesian diagrams: x-axis) value of the marker no. <Mk>, which can (but doesn't have to) be created using `CALCulate<Chn>:MARKer<Mk>[:STATe] ON`.

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace. |
| **<Mk>** | Marker number in the range 1 to 10. |
| **<numeric_value>** | Stimulus value of marker no. <Mk>. |
| Range [def. unit] | –9.9E+11 Hz to +9.9E+11 Hz [Hz] (for frequency sweeps)<br>-999 dBm to +999 dBm [dBm] (for power sweeps)<br>0 s to 127500 s [s] (for time and CW mode sweeps) |
| *RST value | 0 Hz |
| **SCPI, Command Types** | Device-specific, command or query. |
| **Example:** | |

```
Suppose that the active setup 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.

## CALCulate<Chn>:MARKer<Mk>:Y?

Returns the response (in Cartesian diagrams: y-axis) value of marker no. <Mk>. The marker must be created before using `CALCulate<Chn>:MARKer<Mk>[:STATe] ON`.

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace. |
| **<Mk>** | Marker number in the range 1 to 10. |
| **Response** | Response value of marker no. <Mk>. |
| Range [def. unit] | Depending on the measured quantity. [unit depending on the marker format; see `CALCulate<Chn>:MARKer<Mk>:FORMat`] |
| *RST value | – |
| **SCPI, Command Types** | Device-specific, query only. |
| **Example:** | |

```
Suppose that the active setup 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.

## CALCulate<Chn>:MATH...

This subsystem permits processing of measured data in numerical expression format. The operators are +, -, *, / and use of constants and data arrays are permitted.

## 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>:MATH:STATe ON`).

*This command places some restrictions on the mathematical expression and the operands. Use CALCulate<Chn>:MATH[:EXPRession]:SDEFine to define general expressions.*

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace. If unspecified the numeric suffix is set to 1. |
| **<expression>** | (<operand1><operator1><operand2>[<operator2><operand3>])<br>The expression must be enclosed in brackets. |
| Parameters | Operands: See list of trace names in section TRACe<br>Operators: +, -, *, / |
| *RST value | – |
| **SCPI, Command Types** | Confirmed, no query. The defined expression can be queried using CALCulate<Chn>:MATH[:EXPRession]:SDEFine? |
| **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. |

## CALCulate<Chn>:MATH[:EXPRession]:SDEFine '<string>'

Defines a general mathematical relation between traces. To calculate and display the new mathematical trace, the mathematical mode must be switched on (`CALCulate<Chn>:MATH:STATe ON`).

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace. |
| **'<string>'** | String parameter for the mathematical expression, enclosed in brackets. The string format must correspond to an allowed string in the *Define Math* dialog. |

Parameters
Operands, operators and functions; see table below.

*RST value
–

**SCPI, Command Types**
Device-specific, command or query. This command also queries a mathematical expression defined via `CALCulate<Chn>:MATH[:EXPRession][:DEFine]`.

**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 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.

Expressions defined via `CALCulate<Chn>:MATH[:EXPRession]:SDEFine` may contain the following elements:

| Type | Complete List | Description |
|---|---|---|
| **Operands** | <Trace name> \|<br>activeTrc \|<br>Mem[activeTrc] | All traces and memory traces of the active setup \|<br>active trace \|<br>active memory trace assigned to the active trace |
| **Constants** | e, pi \|<br>1, -1.2, 8e9 \|<br>1 + 2j, 2 + 1e-9j | Constants \|<br>Real values in decimal or exponential format \|<br>Complex numbers |
| **Operators** | - + , - , * , / , ^ | Basic arithmetic operations; ^ for exponentiation |
| **Functions** | linMag (), dBMag (), Arg (), Re (), Im (), log (), ln (), tan (), atan (), sin (), asin (), cos (), acos (),<br>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 NORMal | ADD | SUBTract | MULTiply | DIVide

Defines a simple mathematical relation between the active trace and the active memory trace to calculate a new mathematical trace and displays the mathematical trace.

*This command places some restrictions on the mathematical expression and the operands. Use CALCulate<Chn>:MATH[:EXPRession]:SDEFine to define general expressions.*

**<Chn>**
Channel number used to identify the active trace. If unspecified the numeric suffix is set to 1.

**NORMal**
Math. trace = active data trace
**ADD**
Math. trace = data + memory
**SUBTract**
Math. trace = data – memory
**MULTiply**
Math. trace = data * memory
**DIVide**
Math. trace = data / memory

*RST value
NORMal

**SCPI, Command Types** Confirmed, with query.

**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 data 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 switched on, mathematical trace displayed).

## 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 setup 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.

*To copy a trace to the memory without overwriting an existing memory trace or define a memory trace name, use TRACe:COPY <memory_trc>,<data_trc>. To copy an active mathematical trace use TRACe:COPY:MATH <memory_trc>,<data_trc>*

**<Chn>** Channel number used to identify the active trace.

*RST value –

**SCPI, Command Types** Device-specific, no query

**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).

## CALCulate<Chn>:MATH:STATe <Boolean>

Activates or deactivates the mathematical mode where the mathematical trace defined via `CALCulate<Chn>:MATH[:EXPRession]:SDEFine` is calculated and displayed instead of the active

data trace. The command is not valid for mathematical traces calculated via `CALCulate<Chn>:MATH:FUNCtion`.

**<Chn>** Channel number used to identify the active trace.

**<Boolean>** ON - Display the active data trace
OFF - Display the mathematical trace

*RST value OFF

**SCPI, Command Types** Confirmed, command or query.

**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 '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.

## CALCulate<Chn>:MATH:WUNit[:STATe] <Boolean>

Controls the conversion and formatting of the mathematic expression defined via `CALCulate<Chn>:MATH[:EXPRession]:SDEFine` (see *Result is Wave Quantity* in the *User Def Math* dialog).

**<Chn>** Channel number used to identify the active trace.

**<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 value OFF

**SCPI, Command Types** Device-specific, command or query.

**Example:**

```
*RST; SWE:TYPE POW
```

Reset the instrument, activate a power sweep, and select a wave quantity a1 for the trace Trc1.

```
CALC:PAR:SDEF 'Trc1', 'a1'
```

Select a wave quantity a1 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, dividing the data trace by the stored memory trace. Display the mathematical trace 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.

## CALCulate<Ch>:PARameter...

This subsystem assigns names and measurement parameters to traces. The commands are device-specific.

## CALCulate<Ch>:PARameter:CATalog?

Returns the trace names and measurement parameters of all traces assigned to a particular channel.

| | |
|---|---|
| **<Ch>** | Channel number. If unspecified the numeric suffix is set to 1. |
| **Response** | String parameter with 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. |
| *RST value | – |
| **SCPI, Command Types** | Device-specific, query only. |
| **Example:** | `CALC4:PAR:SDEF 'Ch4Tr1', 'S11'` |
| | Create channel 4 and a trace named *Ch4Tr1* to measure the input reflection coefficient S11. |
| | `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'`. |

## CALCulate<Ch>:PARameter:DEFine '<string>', S11 | ... | S44 | A | B | C | D | R1 | R2 | R3 | R4 | AB | AC | AD | BA | BC | BD | CA | CB | CD | DA | DB | DC | AR1 | ... | DR4 | R1A | ... | R4D | R1R2 | ... | R4R3, [<numeric_value>]

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:FEED`); see example below.

Traces must be selected to become active traces; see `CALCulate:PARameter:SELect`.

NOTE

*This command has been implemented for compatibility reasons. The parameter names in this command differ from ZVA conventions; moreover the parameter list is not complete. The alternative command CALCulate<Ch>:PARameter:SDEFine uses a complete parameter list with compatible names.*

| | |
|---|---|
| **<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. |
| **'<string>'** | Trace name, e.g. 'Trc4'. See *Rules for trace names* in the *Trace Manager* description. |
| **S11, ... R4R3** | Measurement parameter; see list of parameters below. |
| *RST value | – |
| **[<numeric_value>]** | Test port number, driving port for wave quantities and ratios, ignored for S-parameters. |
| Range [def. unit] | 1 to 4 [–] (depending on instrument model) |
| *RST value | – |
| **SCPI, Command Types** | Device-specific, no query. `CALCulate<Ch>:PARameter:CATalog?` returns a list of all defined traces. |
| **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:TRAC:FEED 'CH4TR1'` |
| | Display the generated trace in diagram area no. 1. |

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 b1, b2, b3, b4 (received waves) |
| `R1 \| R2 \| R3 \| R4` | Wave quantities a1, a2, a3, a4 (reference waves) |
| `AB \| AC \| AD \| BA \| BC \| BD \| CA \| CB \| CD \| DA \| dB \| DC` | Ratio of wave quantities b1/b2, b1/b3, ... , b4/b3 (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 b1/a1, b1/a2, ... , b4/a4, a1/b1, a1/b2, ..., A4/B4 (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 a1/a2, a1/a3, ... , a4/a3 (reference waves only) |

## CALCulate<Ch>:PARameter:DEFine:SGRoup <log_port1>[, <log_port2>,...]

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:PARameter:SELect`.

**<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.

**<log_port1>...** 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 represent 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 *Trace Manager* (`DISPlay:MENU:KEY:EXECute 'Trace Manager'`)and *Channel 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.

Range [def. unit] 1 to 4 [–] (depending on instrument model)

*RST value NONE (see below)

**SCPI, Command Types** Device-specific, with query. The query returns the port numbers of the current S-parameter group, NONE if no S-parameter group has been defined.

**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 area no. 1.

```
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.

## CALCulate<Ch>:PARameter:DELete '<string>'

Deletes a trace with a specified trace name and channel.

**<Ch>** Channel number.

**'<string>'** Trace name, e.g. 'Trc4'. See *Rules for trace names* in the *Trace Manager* description.

*RST value –

**SCPI, Command Types** Device-specific, no query.

**Example:**

```
CALC4:PAR:SDEF 'Ch4Tr1', 'S11'
```

Create channel 4 and a trace named *Ch4Tr1* to measure the input reflection coefficient S11.

```
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'.

```
CALC4:PAR:DEL 'CH4TR1'
```

Delete the created trace.

## CALCulate<Ch>:PARameter:MEASure '<string>', 'S11' | ... | 'S0101' | ... | 'SCD11' | ... | 'Y11' | ... | 'Z11' | ... | 'Y-S11' | ... | 'Z-S11' | ... | 'A1' | ... | 'A01' | ... | 'B1' | ... | 'B01' | ... | 'B2/A1' | ... 'B02/A01' | ...

Assigns a measurement parameter to an **existing** trace.

***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 window (`DISPlay:WINDow<Wnd>[:STATe] ON`) and assign the trace to this window (`DISPlay:WINDow<Wnd>:TRACe:FEED`); see example below.*

Traces must be selected to become active traces; see `CALCulate:PARameter:SELect`. `CALCulate<Ch>:PARameter:CATalog?` returns a list of all defined traces. You can open the trace manager (`DISPlay:MENU:KEY:EXECute 'Trace Manager'`) to obtain an overview of all channels and traces, including the traces that are not displayed.

**<Ch>** Channel number of an existing channel containing the referenced trace.

**'<string>'** Trace name, e.g. 'Trc4'. See *Rules for trace names* in the *Trace Manager* description.

**'S11', ...** Measurement parameter (string variable); see list of in the `CALCulate<Ch>:PARameter:SDEFine` command description.
A query of a wave quantity 'xy' returns 'xyD<n><Detector>', where <n> numbers the source (drive) port, and <Detector> denotes the detector setting (SAM for a normal (sample), RMS or PEAK for an RMS or peak detector). A query of a ratio 'x/y' returns 'xD<n>/yD<m><Detector>', where <n> and <m> number the source ports.

*RST value –

**SCPI, Command Types** Device-specific, with query (see 'S11',... parameter description above).

**Example:**

```
CALC4:PAR:SDEF 'Ch4Tr1', 'S11'
```

Create channel 4 and a trace named *Ch4Tr1* to measure the input reflection coefficient S11.

```
CALC4:PAR:MEAS 'Ch4Tr1', 'a1'
```

Change the measurement parameter of the trace and measure the wave quantity a1.

```
CALC4:PAR:MEAS? 'Ch4Tr1'
```

Query the measured quantity. The response is 'A1D1SAM'.

## CALCulate<Ch>:PARameter:SDEFine '<trc_name>', 'S11' | ... | 'S0101' | ... | 'SCD11' | ... | 'Y11' | ... | 'Z11' | ... | 'Y-S11' | ... | 'Z-S11' | ... | 'A1' | ... | 'A01' | ... | 'B1' | ... | 'B01' | ... | 'B2/A1' | ... 'B02/A01' | ... | 'Pmtr2D1' | ...

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 area (`DISPlay:WINDow<Wnd>[:STATe] ON`) and assign the trace to this area (`DISPlay:WINDow<Wnd>:TRACe:FEED`); see example below. `CALCulate<Ch>:PARameter:MEASure` changes the measurement parameter of an existing trace.*

To select an existing trace as the active trace, use `CALCulate:PARameter:SELect`. You can open the trace manager (`DISPlay:MENU:KEY:EXECute 'Trace Manager'`) to obtain an overview of all channels and traces, including the traces that are not displayed.

NOTE *In FW versions <V1.90, traces created with `CALCulate<Ch>:PARameter:SDEFine` must be selected explicitly (`CALCulate:PARameter:SELect`) in order to become active traces.*

TIP Use `CALCulate<Ch>:PARameter:DEFine:SGRoup` to create traces for all S-parameters associated with a group of logical ports.

**<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.

**'<trc_name>'** Trace name, string variable, e.g. 'Trc4'. See *Rules for trace names* in the *Trace Manager* description.
Trace names must be unique across all channels and diagram areas. 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.

**'S11', ...**
Measurement parameter (string variable); see list of parameters below.

*RST value
–

**SCPI, Command Types**
Device-specific, no query. `CALCulate<Ch>:PARameter:MEASure? '<Trc_name>'` queries the measurement parameter of the trace. `CALCulate<Ch>:PARameter:CATalog?` returns a list of all defined traces.

**Example:**

```
CALC4:PAR:SDEF 'Ch4Tr1', 'S11'
```

Create channel 4 and a trace named *Ch4Tr1* to measure the input reflection coefficient S11.

```
DISP:WIND2:STAT ON
```

Create diagram area no. 2.

```
DISP:WIND2:TRAC:FEED 'CH4TR1'
```

Display the generated trace in diagram area no. 2.

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' \| ... \| 'S0101' \| ...` | Normal mode S-parameters S<out><in>, where <out> and <in> denote the output and input port numbers of the DUT. To avoid ambiguities, <out> and <in> must be either both 1-digit numbers (e.g. 21) or both 2-digit numbers (e.g. 2121 or 0133). |
| `'SCD11' \|` | Mixed mode S-parameters S<mout><m in>, where <mout> and <min> denote the output and input port modes. All combinations of S (single-ended, unbalanced), D (differential) and C (common) are allowed. Port numbers like for normal mode S-parameters. |
| `'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*). |
| `'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. |
| `'A1' \| ... \| 'A01' \| ... \| 'B1' \| ... \| 'B01' \| ... \| 'A1SAM' \| 'A1RMS' \| 'A1PEAK' \|` | Wave quantities with port numbers like for normal mode S-parameters. The strings SAM, RMS, PEAK appended to the wave quantities denote a normal (sample), RMS or peak detector. The observation time for RMS and peak detectors is set via `[SENSe<Ch>:]SWEep:DETector:TIME`. |
| `'A1D2' \| ... \| 'A01D2' \| ... \| 'B1D2' \| ... \| 'B01D2' \| ... \| 'A1D1SAM' \| 'A1D1RMS' \| 'A1D1PEAK' \| ...` | Wave quantities with port numbers and source port numbers (D<no> for drive port). The strings SAM, RMS, PEAK appended to the wave quantities denote a normal (sample), RMS or peak detector. |
| `'A1G2' \| ... \| 'A01G2' \| ... \| 'B1G2' \| ... \| 'B01G2' \| ... \| 'A1G1SAM' \| 'A1G1RMS' \| 'A1G1PEAK' \| ...` | Wave quantities with port numbers and external generator providing the stimulus signal (G<no> for generator no.). The strings SAM, RMS, PEAK appended to the wave quantities denote a normal (sample), RMS or peak detector. |
| `'AS1D2S' \| ... \| 'AS1D2C' \| ... 'BS1D2S' \| 'AS1D2SSAM' \| 'AS1D2SRMS' \| 'AS1D2SPEAK' \| ...` | Balanced (D, C) or single-ended (S) wave quantities with port numbers and source port numbers (**D**<no> for drive port) in true differential mode (option R&S ZVA-K6). The strings SAM, RMS, PEAK appended to the wave quantities denote a normal (sample), RMS or peak detector. |
| `'B2/A1' \| ... 'B02/A01' \| ... \| 'B2/A1SAM' \| 'B2/A1RMS' \| 'B2/A1PEAK' \| ...` | Ratio of wave quantities with port numbers like for normal mode S-parameters. The strings SAM, RMS, PEAK appended to the wave quantities denote a normal (sample), RMS or peak detector. |

| | |
|---|---|
| 'B2D1/A1D1' \| ... 'B02D1/A01D1' \| ... 'B2D1/A1D1SAM' \| 'B2D1/A1D1RMS' \| 'B2D1/A1D1PEAK' \| ... | Ratio of wave quantities with port numbers and source port numbers (D<no> for drive port). The strings SAM, RMS, PEAK appended to the wave quantities denote a normal (sample), RMS or peak detector. |
| 'B2G1/A1G1' \| ... 'B02G1/A01G1' \| ... 'B2G1/A1G1SAM' \| 'B2G1/A1G1RMS' \| 'B2G1/A1G1PEAK' \| ... | Ratio of wave quantities with port numbers and external generator providing the stimulus signal (G<no> for generator no.). The strings SAM, RMS, PEAK appended to the wave quantities denote a normal (sample), RMS or peak detector. |
| 'BD1**D**1D/AD1**D**1D' \| ... \| BD1**D**1D/AD1**D**3S' \| 'BD1**D**1D/AD1**D**3SSAM' \| 'BD1**D**1D/AD1**D**3SRMS' \| 'BD1**D**1D/AD1**D**3SPEAK' \| ... | Ratio of balanced (D, C) or single-ended (S) wave quantities with port numbers and source port numbers (**D**<no> for drive port) in true differential mode (option R&S ZVA-K6). The strings SAM, RMS, PEAK appended to the wave quantities denote a normal (sample), RMS or peak detector. |
| 'IMB21' \| 'IMB12' \| 'IMB31' \| ... | Imbalance parameter for balanced ports Imb<receive_port><drive_port>. The indices denote the logical receive port and the logical drive port of the analyzer. |
| 'KFAC21' \| 'KFAC12' \| ... | Stability factor K (for unbalanced ports only) |
| 'MUF121' \| 'MUF112' \| ... | Stability factor $\mu_1$ (for unbalanced ports only) |
| 'MUF221' \| 'MUF212' \| ... | Stability factor $\mu_2$ (for unbalanced ports only) |
| 'PAE21' \| PAE12' \| | Power added efficiency |
| 'DC1V' \| 'DC10V' (query returns 'DC ± 1V' or 'DC ± 10V') | DC MEAS ± 1 V, DC MEAS ± 10 V |
| 'Pmtr1G1' \| 'Pmtr2G1' \| 'Pmtr3D1' | Power sensor measurement using a power meter 'Pmtr<n>' and either an external generator 'G<n>' or an analyzer source port 'D<n>' |

*)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:SELect <string>

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<Ch>:FORMat`). `CALCulate<Ch>:PARameter:SELect` is also necessary if the active trace of a channel has been deleted.

**<Ch>** Channel number.

**<string>** Trace name, e.g. 'Trc4'. See *Rules for trace names* in the *Trace Manager* description.

*RST value –

**SCPI, Command Types** Device-specific, with query (returns the name of the active trace).

**Example:**

```
CALC4:PAR:SDEF 'Ch4Tr1', 'S11'
```

Create channel 4 and a trace named *Ch4Tr1* to measure the input reflection coefficient S11. 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 S22. 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 S11 and display it in a linearly scaled Cartesian diagram.

## CALCulate<Chn>:PHOLd

This subsystem controls the max hold (peak hold) function.

## CALCulate<Chn>:PHOLd MAX | OFF

Enables, disables, or restarts the max hold function.

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace. |
| **Parameters** | MAX - Enable the max hold function<br>OFF - Disables the max hold function |
| *RST value | OFF |
| **SCPI, Command Types** | Device-specific, command or query. |
| **Example:** | `*RST; CALC:PHOL MAX` |
| | Reset the instrument and enable the max hold function. |
| | `CALC:PHOL OFF; PHOL MAX` |
| | Restart max hold. |

## CALCulate<Chn>:STATistics...

This subsystem evaluates and displays statistical and phase information of the trace.

## CALCulate<Chn>:STATistics:DOMain:USER <numeric_value>

Selects one out of 10 evaluation ranges to be configured with the `CALCulate<Chn>:STATistics:DOMain:USER:STARt` and `CALCulate<Chn>:STATistics:DOMain:USER:STOP` commands.

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace. If unspecified the numeric suffix is set to 1. |
| **<numeric_value>** | Number of the evaluation range. |
| Range [def. unit] | 1 to 10. In addition, 0 denotes the (non-configurable) *Full Span* evaluation range. [–] |

| | |
|---|---|
| *RST value | 0 |
| **SCPI, Command Types** | Device-specific, command or query. |
| **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` |
| | Select evaluation range no. 1 and define the evaluation range between 1 GHz and 2 GHz. |

## CALCulate<Chn>:STATistics:DOMain:USER:STARt <numeric_value>

Defines the start value of the evaluation range selected via `CALCulate<Chn>:STATistics:DOMain:USER`.

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace. |
| **<numeric_value>** | Start value of the evaluation range. |
| Range [def. unit] | –1000 GHz to 999.999999999 GHz. [Hz] |
| *RST value | Lowest frequency of the analyzer, depending on the analyzer model. |
| **SCPI, Command Types** | Device-specific, command or query. |
| **Example:** | `See CALCulate<Chn>:STATistics:DOMain:USER` |

## CALCulate<Chn>:STATistics:DOMain:USER:STOP <numeric_value>

Defines the stop value of the evaluation range selected via `CALCulate<Chn>:STATistics:DOMain:USER`.

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace. |
| **<numeric_value>** | Stop value of the evaluation range. |
| Range [def. unit] | –999.999999999 GHz to 1000 GHz. [Hz] |
| *RST value | Highest frequency of the analyzer, depending on the analyzer model. |
| **SCPI, Command Types** | Device-specific, command or query. |
| **Example:** | `See CALCulate<Chn>:STATistics:DOMain:USER` |

## CALCulate<Chn>:STATistics:EPDelay[:STATe] <Boolean>

Displays or hides the *Phase Delay/El Length* results in the diagram area of trace no. <Chn>.

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace. |
| **<Boolean>** | ON \| OFF - Statistical info field on or off. |
| *RST value | OFF |

| | |
|---|---|
| **SCPI, Command Types** | Device-specific, command or query. |
| **Example:** | See `CALCulate<Chn>:STATistics:[:STATe]` |

### CALCulate<Chn>:STATistics:MMPTpeak[:STATe] <Boolean>

Displays or hides the *Min/Max/Peak-Peak* results in the diagram area of trace no. <Chn>.

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace. |
| **<Boolean>**<br>*RST value | ON \| OFF - Statistical info field on or off.<br>OFF |
| **SCPI, Command Types** | Device-specific, command or query. |
| **Example:** | See `CALCulate<Chn>:STATistics:[:STATe]` |

### CALCulate<Chn>:STATistics:MSTDev[:STATe] <Boolean>

Displays or hides the *Mean/Std Dev* results in the diagram area of trace no. <Chn>.

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace. |
| **<Boolean>**<br>*RST value | ON \| OFF - Statistical info field on or off.<br>OFF |
| **SCPI, Command Types** | Device-specific, command or query. |
| **Example:** | See `CALCulate<Chn>:STATistics:[:STATe]` |

### CALCulate<Chn>:STATistics:NLINear:COMP:LEVel <numeric_value>

Defines the compression value x for the x-dB compression point measurement (`CALCulate<Chn>:NLINear:COMP:RESult?`).

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace. |
| **<numeric_value>**<br>Range [def. unit]<br>*RST value | Stop value of the evaluation range.<br>+0.01 dBm to +100.00 dBm [dBm]<br>1 dBm |
| **SCPI, Command Types** | Device-specific, command or query. |
| **Example:** | See `CALCulate<Chn>:NLINear:COMP:RESult?` |

### CALCulate<Chn>:STATistics:NLINear:COMP:RESult?

Returns the x-dB compression point of an S-parameter or ratio measured in a power sweep. The compression value x is set via `CALCulate<Chn>:STATistics:NLINear:COMP:LEVel`.

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace. |

| | |
|---|---|
| **Result** | <Cmp_In>, <Cmp_Out><br><Cmp_In> is the stimulus level at the compression point<br><Cmp_Out> is the magnitude of the wave reflected or transmitted by the DUT. |
| Range [def. unit] | For <Cmp_In>: Current sweep range or evaluation range (CALCulate<Chn>:STATistics:DOMain:USER), if different from the sweep range. [dBm]<br>For <Cmp_Out>: Measurement range of the analyzer. [dBm] |
| *RST value | – |
| **SCPI, Command Types** | Device-specific, query only. |
| **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. |

## CALCulate<Chn>:STATistics:NLINear:COMP[:STATe] <Boolean>

Displays or hides the compression point result in the diagram area of trace no. <Chn>.

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace. |
| **<Boolean>**<br>*RST value | ON \| OFF - Statistical info field on or off.<br>OFF |
| **SCPI, Command Types** | Device-specific, command or query. |
| **Example:** | See `CALCulate<Chn>:NLINear:COMP:RESult?` |

## CALCulate<Chn>:STATistics:RESult? MEAN | STDDev | MAX | MIN | RMS | PTPeak | ELENgth | PDELay | GAIN | SLOPe | FLATness | ALL

Returns a single statistical parameters 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.

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace. |
| **Parameters** | 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)<br>STDDev – Return standard deviation of all response values<br>MAX – Return the maximum of all response values<br>MIN – Return the minimum of all response values<br>RMS – Return the root mean square of all response values |

| | |
|---|---|
| | PTPeak – Return the peak-to-peak value (MAX – MIN)<br>ELENgth – Return the electrical length<br>PDELay – Return the phase delay<br>GAIN – Return the gain, i.e. the larger of two marker values<br>SLOPe – Return the slope (difference) between two marker values<br>FLATness – Return the flatness of the trace between two marker positions<br>ALL – Return all statistical values, observing the order used above |
| Range [def. unit] | 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 also depends on the trace format (linear or logarithmic scale, see `CALCulate<Chn>:FORMat)`. If a polar trace format is selected, then the statistical parameters are calculated from the linear magnitude of the measurement parameter. [–] |
| *RST value | – |
| **SCPI, Command Types** | Device-specific, query only. |
| **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. |

## CALCulate<Chn>:STATistics:RMS[:STATe] <Boolean>

Displays or hides the *RMS* results in the diagram area of trace no. <Chn>.

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace. |
| **<Boolean>**<br>*RST value | ON \| OFF - Statistical info field on or off.<br>OFF |
| **SCPI, Command Types** | Device-specific, command or query. |
| **Example:** | See `CALCulate<Chn>:STATistics:[:STATe]` |

## CALCulate<Chn>:STATistics:SFLatness[:STATe] <Boolean>

Displays or hides the *Gain/Slope/Flatness* results in the diagram area of trace no. <Chn>.

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace. |
| **<Boolean>**<br>*RST value | ON \| OFF - Statistical info field on or off.<br>OFF |
| **SCPI, Command Types** | Device-specific, command or query. |
| **Example:** | See `CALCulate<Chn>:STATistics:[:STATe]` |

## CALCulate<Chn>:STATistics[:STATe] <Boolean>

Displays or hides all statistical results in the diagram area of trace no. <Chn> except the compression point results.

You can display or hide the *Min/Max/Peak-Peak, Mean/Std Dev, RMS, Phase Delay/El Length* and *Gain/Slope/Flatness* results separately; see example below.

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace. |
| **<Boolean>**<br>*RST value | ON \| OFF - Statistical info field on or off.<br>OFF |
| **SCPI, Command Types** | Device-specific, command or query. |
| **Example:** | `*RST; CALC:STAT:MMPT ON`<br>Reset the instrument, hiding all statistical results. Display the *Min/Max/Peak-Peak* results.<br>`CALC:STAT:MSTD ON`<br>Display the *Mean/Std Dev* results in addition.<br>`CALC:STAT:RMS ON`<br>Display the *RMS* results in addition.<br>`CALC:STAT:EPD ON`<br>Display the *Phase Delay/El Length* results in addition.<br>`CALC:STAT:SFL ON`<br>Display the *Gain/Slope/Flatness* results in addition.<br>`CALC:STAT OFF`<br>Hide all results. |

## CALCulate<Chn>:SMOothing...

This subsystem provides the settings for trace smoothing.

## CALCulate<Chn>:SMOothing:APERture <numeric_value>

Defines how many measurement points are averaged to smooth the trace.

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace. If unspecified the numeric suffix is set to 1. |
| **<numeric_value>**<br>Range [def. unit] | Smoothing aperture.<br>0.05% to 100%. 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. [%]

*RST value 1

**SCPI, Command Types** Device-specific, command or query.

**Example:**

```
*RST; CALC:SMO ON
```

Activate smoothing for the default trace.

```
CALC:SMO:APER 0.5
```

Reduce the smoothing aperture to 0.5 %.

## CALCulate<Chn>:SMOothing[:STATe] <Boolean>

Enables or disables smoothing for trace no. <Chn>.

**<Chn>** Channel number used to identify the active trace.

**<Boolean>** ON | OFF - Smoothing on or off.
*RST value OFF

**SCPI, Command Types** Device-specific, command or query.

**Example:**

```
See CALCulate<Chn>:STATistics:SMOothing:APERture
```

## CALCulate<Ch>:TDIF...

This subsystem provides settings for the *True Differential Mode* (with option R&S ZVA-K6).

## CALCulate<Ch>:TDIF:IMBalance:COMPensation[:STATe] <Boolean>

Selects the calculation method for S-parameters, ratios and derived quantities during an amplitude imbalance or phase imbalance sweep.

**<Ch>** Channel number

**<Boolean>** ON – Compensate imbalance of a waves
OFF – Use the actual a-waves, depending on the imbalance parameters

*RST value OFF [–]

**SCPI, Command Types** Device-specific, command or query

**Example:**

```
See SOURce<Ch>:TDIF:IMBalance:AMPLitude:LPORt
```

## CALCulate<Chn>:TRANsform...

This subsystem converts measured data from one representation to another and controls the transformation into the time domain (with option ZVAB-K2, *Time Domain)*.

## CALCulate<Chn>:TRANsform:COMPlex S | Y | Z

Converts S-parameters into matched-circuit (converted) 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:

$$Z_{<out><in>ii} = Z_0 \frac{1+S_{<out><in>ii}}{1-S_{<out><in>ii}}, \quad <out> = <in>,$$

$$Z_{<out><in>ij} = 2 \cdot \frac{\sqrt{Z_{0i} \cdot Z_{0j}}}{S_{<out><in>ij}} - (Z_{0i} + Z_{0j}), \quad i \neq j,$$

$$Y_{ii} = \frac{1}{Z_{0i}} \frac{1-S_{ii}}{1+S_{ii}} = 1/Z_{ii},$$

$$Y_{ij} = \frac{S_{ij}}{2 \cdot \sqrt{Z_{0i} \cdot Z_{0j}} - S_{ij} \cdot (Z_{0i} + Z_{0j})} = 1/Z_{ij}, \quad i \neq j,$$

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace. If unspecified the numeric suffix is set to 1. |
| **S \| Y \| P**<br>*RST value | S-parameters, Y-parameters, Z-parameters<br>– (The initial representation of the trace is determined by means of `CALCulate<Ch>:PARameter:SDEFine`) |
| **SCPI, Command Types** | Device-specific, command or query. |
| **Example:** | `*RST; CALC:PAR:MEAS 'Trc1'", '"Y-S22'`<br>Select the converted admittance Y<–S22 as measurement parameter of the default trace.<br>`CALC:TRAN:COMP S`<br>Convert the converted Y-parameter into an S-parameter. |

## CALCulate<Chn>:TRANsform:TIME:CENTer <numeric_value>

Defines the center time of the diagram in time domain.

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace |
| **<numeric_value>**<br>Range [def. unit] | Center time of the diagram in time domain<br>-99.99999999999 s to +99.99999999999 s. The increment (parameters |

| | |
|---|---|
| | UP or DOWN) is 0.1 ns. [s] |
| *RST value | 1.5E-009 s |
| **SCPI, Command Types** | Confirmed, command or query |
| **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. |

***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 <numeric_value>

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`).

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace |
| **<numeric_value>** | Sideband suppression |
| Range [def. unit] | 10 dB to 120 dB. The increment (parameters UP or DOWN) is 10 dB. [dB] |
| *RST value | 32 dB |
| **SCPI, Command Types** | Device-specific, command or query |
| **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. |

## CALCulate<Chn>:TRANsform:TIME:LPASs KFSTop | KDFRequency | KSDfrequency

Calculates the harmonic grid for low pass time domain transforms according to one of the three alternative algorithms.

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace |
| **Parameters** | KFSTop – Keep stop frequency and number of points<br>KDFRequency – Keep frequency gap and number of points<br>KSDfrequency – Keep stop frequency and approximate frequency gap |
| *RST value | – |
| **SCPI, Command Types** | Device-specific, no query (the command causes an event) |
| **Example:** | `See CALCulate<Chn>:TRANsform:TIME[:TYPE]` |

## CALCulate<Chn>:TRANsform:TIME:LPASs:DCSParam <numeric_value>

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`).

**<Chn>** Channel number used to identify the active trace

**<numeric_value>** DC value of the measured quantity
Range [def. unit] Depending on the measured quantity (–1 to +1 for S-parameters)
*RST value 0

**SCPI, Command Types** Device-specific, command or query

**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, keeping 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.

## CALCulate<Chn>:TRANsform:TIME:LPASs:DCSParam:CONTinuous <Boolean>

Determines whether continuous extrapolation for the DC value is enabled.

**<Chn>** Channel number used to identify the active trace

**<Boolean>** ON – Continuous extrapolation enabled
OFF – Continuous extrapolation disabled
*RST value OFF [–]

**SCPI, Command Types** Device-specific, command or query

**Example:**

```
See CALCulate<Chn>:TRANsform:TIME:LPASs:DCSParam
```

## 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.

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace |
| **Parameters**<br>*RST value | None<br>– |
| **SCPI, Command Types** | Device-specific, no query (the command causes an event) |
| **Example:** | `See CALCulate<Chn>:TRANsform:TIME:LPASs:DCSParam` |

## CALCulate<Chn>:TRANsform:TIME:LPFRequency

Calculates the harmonic grid for low pass time domain transforms, keeping the stop frequency and the number of points.

Use `CALCulate<Chn>:TRANsform:TIME:LPASs` if you wish to use one of the other algorithms for calculating the grid.

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace |
| **Parameters**<br>*RST value | None<br>– |
| **SCPI, Command Types** | Device-specific, no query (the command causes an event) |
| **Example:** | `See CALCulate<Chn>:TRANsform:TIME[:TYPE]` |

## CALCulate<Chn>:TRANsform:TIME:RESolution:EFACtor <factor>

Defines the resolution enhancement factor for the time domain transform.

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace |
| **<factor>**<br>Range [def. unit]<br>*RST value | Resolution enhancement factor.<br>1 to 10. The increment (parameters UP or DOWN) is 0.1. [–]<br>1 (no resolution enhancement) |
| **SCPI, Command Types** | Device-specific, command or query |
| **Example:** | `See CALCulate<Chn>:TRANsform:TIME:LPASs:DCSParam` |

## CALCulate<Chn>:TRANsform:TIME:SPAN <time_span>

Defines the time span of the diagram in time domain.

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace |
| **<time_span>** | Time span of the diagram in time domain. |

| | |
|---|---|
| Range [def. unit] | 2E-012 s to 200 s. The increment (parameters UP or DOWN) is 0.1 ns. [s] |
| *RST value | 5E-009 s |
| **SCPI, Command Types** | Confirmed, command or query |
| **Example:** | `See CALCulate<Chn>:TRANsform:TIME:CENTer` |

***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 <start_time>

Defines the start time of the diagram in time domain.

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace |
| **<start_time>** | Start time of the diagram. |
| Range [def. unit] | –100 s to +99.999999999998 s. The increment (parameters UP or DOWN) is 0.1 ns. [s] |
| *RST value | –1E-009 s |
| **SCPI, Command Types** | Confirmed, command or query |
| **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. |

***Note:*** *If the start frequency entered is greater than the current stop frequency (`CALCulate<Chn>:TRANsform:TIME:STOP`), the stop frequency is set to the start frequency plus the minimum frequency span (`CALCulate<Chn>:TRANsform:TIME:SPAN`).*
*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.

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace |
| **<Boolean>** | ON – Time domain representation active<br>OFF – Frequency domain representation active |
| *RST value | OFF [–] |
| **SCPI, Command Types** | Confirmed, command or query |
| **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*. |

## CALCulate<Chn>:TRANsform:TIME:STIMulus STEP | IMPulse

Selects the type of stimulus to be simulated in the low pass transformation process.

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace |
| **Parameters** | IMPulse – Impulse response, in bandpass or lowpass mode<br>STEP – Step response, only in lowpass mode (a bandpass mode setting `CALCulate<Chn>:TRANsform:TIME[:TYPE] BPASs` is automatically changed to lowpass) |
| *RST value | IMP |
| **SCPI, Command Types** | Confirmed, command or query |
| **Example:** | `See CALCulate<Chn>:TRANsform:TIME[:TYPE]` |

## CALCulate<Chn>:TRANsform:TIME:STOP <numeric_value>

Defines the stop time of the diagram in time domain.

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace |
| **<numeric_value>** | Stop time of the diagram. |
| Range [def. unit] | –99.999999999998 s to +100 s. The increment (parameters UP or DOWN) is 0.1 ns. [s] |
| *RST value | +4E-009 s |
| **SCPI, Command Types** | Confirmed, command or query |
| **Example:** | `See CALCulate<Chn>:TRANsform:TIME:STARt.` |

***Note:*** *If the stop frequency entered is smaller than the current start frequency (`CALCulate<Chn>:TRANsform:TIME:STARt`), the start frequency is set to the stop frequency minus the minimum frequency span (`CALCulate<Chn>:TRANsform:TIME:SPAN`). 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] BPASs | LPASs

Selects the time domain transformation type.

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace |
| **Parameters** | BPASs – Band pass impulse (only impulse response; a step response `CALCulate<Chn>:TRANsform:TIME:STIMulus STEP` is automatically changed to impulse response)<br>LPASs – Low pass (impulse or step response, depending on `CALCulate<Chn>:TRANsform:TIME:STIMulus` setting) |
| *RST value | BPASs |
| **SCPI,** | Confirmed, command or query |

**Command Types**

**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.

## CALCulate<Chn>:TRANsform:TIME:WINDow RECT | HAMMing | HANN | BOHMan | DCHebyshev

Selects the window type for filtering the data in the frequency domain prior to the time domain transformation.

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace |
| **Parameters** | RECT – No profiling (rectangle)<br>HANN – Normal profile (Hann)<br>HAMMing – Low first sidelobe (Hamming)<br>BOHMan – Steep falloff (Bohman)<br>DCHebyshev – Arbitrary sidelobes (Dolph-Chebychev) |
| *RST value | HANN |
| **SCPI, Command Types** | Confirmed (with some device-specific parameters), command or query |
| **Example:** | `See CALCulate<Chn>:TRANsform:TIME:DCHebychev` |

## CALCulate<Chn>:TRANsform:TIME:XAXis TIME | DISTance

Switches over between the x-axis scaling in time units or distance units.

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace |
| **TIME** | x-axis scaled in time units |
| **DISTance** | x-axis scaled in distance units *(Distance = Time * $c_0$ * Velocity Factor)* |
| *RST value | OFF [–] |
| **SCPI, Command Types** | Device-specific, command or query |
| **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.

## CALCulate<Ch>:TRANsform:VNETworks...

This subsystem defines the circuit models for single ended and balanced port (de-)embedding and activates the (de-)embedding function.

The circuit models are referenced by means of predefined character data parameters. They are different for single ended and balanced port (de-)embedding.

**Circuit models for balanced port embedding/deembedding:**

| Parameter | Circuit model | Capacitances | Resistances | Inductances |
|---|---|---|---|---|
| FIMPort | File import, no circuit model | – | – | – |
| STSL | Serial Touchstone (.s2p) data, shunt L | – | R1 | L1 |
| STSC | Serial Touchstone (.s2p) data, shunt C | C1 | R1 | – |
| SLST | Shunt L, serial Touchstone (.s2p) data | – | R1 | L1 |
| SCST | Shunt C, serial Touchstone (.s2p) data | C1 | R1 | – |
| CSSL | Serial Cs, shunt L | C1, C2 | R1, R2, R3 | L1 |
| LSSC | Serial Ls, shunt C | C1 | R1, R2, R3 | L1, L2 |
| CSSC | Serial Cs, shunt C | C1, C2, C3 | R1, R2, R3 | – |
| LSSL | Serial Ls, shunt L | – | R1, R2, R3 | L1, L2, L3 |
| SLCS | Shunt L, serial Cs | C1, C2 | R1, R2, R3 | L1 |
| SCLS | Shunt C, serial Ls | C1 | R1, R2, R3 | L1, L2 |
| SCCS | Shunt C, serial Cs | C1, C2, C3 | R1, R2, R3 | – |
| SLLS | Shunt L, serial Ls | – | R1, R2, R3 | L1, L2, L3 |

**Circuit models for single ended port embedding/deembedding:**

| Parameter | Circuit model | Capacitances | Resistances | Inductances |
|---|---|---|---|---|
| FIMPort | File import, no circuit model | – | – | – |

| | | | | |
|---|---|---|---|---|
| CSL | Serial C, shunt L | C1 | R1, R2 | L1 |
| LSC | Serial L, shunt C | C1 | R1, R2 | L1 |
| CSC | Serial C, shunt C | C1, C2 | R1, R2 | – |
| LSL | Serial L, shunt L | – | R1, R2 | L1, L2 |
| SLC | Shunt L, serial C | C1 | R1, R2 | L1 |
| SCL | Shunt C, serial L | C1 | R1, R2 | L1 |
| SCC | Shunt C, serial C | C1, C2 | R1, R2 | – |
| SLL | Shunt L, serial L | – | R1, R2 | L1, L2 |

## CALCulate<Ch>:TRANsform:VNETworks:BALanced:DEEMbedding<Log_pt>:PARameters:C<no> STSC | SCST | CSSL | LSSC | CSSC | SLCS | SCLS | SCCS,<numeric_value>

Specifies the capacitance values C1, C2, C3 in the different circuit models for balanced port deembedding.

**<Ch>** Channel number. If unspecified the numeric suffix is set to 1.

**<Log_pt>** Logical port number

**<no>** Number of capacitance in circuit model. If unspecified the numeric suffix is set to 1.

Range [def. unit] 1 to 3 [–] The number of capacitances depends on the selected circuit model.

**Parameters** Possible circuit models (character data); see circuit models.

*RST value – (see command type description below)

**<numeric_value>** Capacitance C<no> for the specified circuit model.

Range [def. unit] –1mF to 1 mF [F]. The increment is 1 fF ($10^{-15}$ F).

*RST value 1 pF ($10^{-12}$ F)

**SCPI, Command Types** Device-specific, command or query. In the query form, the <numeric_value> parameter must be omitted. The command returns the capacitance value for the specified circuit model.

**Example:**

```
*RST; CALC:TRAN:VNET:BAL:DEEM:PAR:C2? CSSL
```

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.

## CALCulate<Ch>:TRANsform:VNETworks:BALanced:DEEMbedding<Log_pt>:PARameters:L<no> STSL | SLST | CSSL | LSSC | LSSL | SLCS | SCLS |

## SLLS,<numeric_value>

Specifies the inductance values L1, L2, L3 in the different circuit models for balanced port deembedding.

| | |
|---|---|
| **<Ch>** | Channel number |
| **<Log_pt>** | Logical port number |
| **<no>** | Number of inductance in circuit model. If unspecified the numeric suffix is set to 1. |
| Range [def. unit] | 1 to 3 [–] The number of inductances depends on the selected circuit model. |
| **Parameters** | Possible circuit models (character data); see circuit models. |
| *RST value | – (see command type description below) |
| **<numeric_value>** | Inductance L<no> for the specified circuit model. |
| Range [def. unit] | –1H to 1 H [H]. The increment is 1 pH ($10^{-12}$ H). |
| *RST value | 1 nH ($10^{-9}$ H) |
| **SCPI, Command Types** | Device-specific, command or query. In the query form, the <numeric_value> parameter must be omitted. The command returns the inductance value for the specified circuit model. |

**Example:**

```
*RST; CALC:TRAN:VNET:BAL:DEEM:PAR:L1? CSSL
```

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.

## CALCulate<Ch>:TRANsform:VNETworks:BALanced:DEEMbedding<Log_pt>:PARameters:R<no> STSL | STSC | SLST | SCST | CSSL | LSSC | CSSC | LSSL | SLCS | SCLS | SCCS | SLLS,<numeric_value>

Specifies the resistance values R1, R2, R3 in the different circuit models for balanced port deembedding.

| | |
|---|---|
| **<Ch>** | Channel number |
| **<Log_pt>** | Logical port number |
| **<no>** | Number of resistance in circuit model. If unspecified the numeric suffix is set to 1. |
| Range [def. unit] | 1 to 3 [–] The number of resistances depends on the selected circuit model. |
| **Parameters** | Possible circuit models (character data); see circuit models. |
| *RST value | – (see command type description below) |
| **<numeric_value>** | Resistance R<no> for the specified circuit model. |
| Range [def. unit] | –10 MΩ to 10 MΩ [Ω]. The increment is 1 mΩ ($10^{-3}$ Ω). |
| *RST value | 0 Ω for all resistances connected in series with an inductance<br>10 MΩ for all resistances connected in parallel with a capacitance |
| **SCPI, Command Types** | Device-specific, command or query. In the query form, the <numeric_value> parameter must be omitted. The command returns the resistance value for the specified circuit model. |

**Example:**

```
*RST; CALC:TRAN:VNET:BAL:DEEM:PAR:R1? CSSL; R2? CSSL, R3? CSSL
```

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.

## CALCulate<Ch>:TRANsform:VNETworks:BALanced:DEEMbedding<Log_pt>[: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.

| | |
|---|---|
| **<Ch>** | Channel number |
| **<Log_pt>** | Logical port number |
| **<Boolean>** | ON – Deembedding active<br>OFF – Deembedding inactive |
| *RST value | OFF [–] |
| **SCPI, Command Types** | Device-specific, command or query |

**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.

## CALCulate<Ch>:TRANsform:VNETworks:BALanced:DEEMbedding<Log_pt>:TNDefinition FIMPort | STSL | STSC | SLST | SCST | CSSL | LSSC | CSSC | LSSL | SLCS | SCLS | SCCS | SLLS

Selects the circuit model for balanced port deembedding.

| | |
|---|---|
| **<Ch>** | Channel number |
| **<Log_pt>** | Logical port number |
| **Parameters** | Possible circuit models (character data); see circuit models. |
| *RST value | CSSL |
| **SCPI, Command Types** | Device-specific, command or query. |

**Example:**

```
See CALCulate<Ch>:TRANsform:VNETworks:BALanced:DEEMbedding[:STATe]
```

## CALCulate<Ch>:TRANsform:VNETworks:BALanced:EMBedding<Log_pt>:PARameters:C<no> STSC | SCST | CSSL | LSSC | CSSC | SLCS | SCLS | SCCS,<numeric_value>

Specifies the capacitance values C1, C2, C3 in the different circuit models for balanced port embedding.

| | |
|---|---|
| **<Ch>** | Channel number |
| **<Log_pt>** | Logical port number |
| **<no>** | Number of capacitance in circuit model. If unspecified the numeric suffix is set to 1. |
| Range [def. unit] | 1 to 3 [–] The number of capacitances depends on the selected circuit model. |
| **Parameters** | Possible circuit models (character data); see circuit models. |
| *RST value | – (see command type description below) |
| **<numeric_value>** | Capacitance C<no> for the specified circuit model. |
| Range [def. unit] | –1mF to 1 mF [F]. The increment is 1 fF ($10^{-15}$ F). |
| *RST value | 1 pF ($10^{-12}$ F) |
| **SCPI, Command Types** | Device-specific, command or query. In the query form, the <numeric_value> parameter must be omitted. The command returns the capacitance value for the specified circuit model. |
| **Example:** | `*RST; CALC:TRAN:VNET:BAL:EMB:PAR:C2? CSSL` |
| | 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. |

## CALCulate<Ch>:TRANsform:VNETworks:BALanced:EMBedding<Log_pt>:PARameters:L<no> STSL | SLST | CSSL | LSSC | LSSL | SLCS | SCLS | SLLS,<numeric_value>

Specifies the inductance values L1, L2, L3 in the different circuit models for balanced port embedding.

| | |
|---|---|
| **<Ch>** | Channel number |
| **<Log_pt>** | Logical port number |
| **<no>** | Number of inductance in circuit model. If unspecified the numeric suffix is set to 1. |
| Range [def. unit] | 1 to 3 [–] The number of inductances depends on the selected circuit model. |
| **Parameters** | Possible circuit models (character data); see circuit models. |
| *RST value | – (see command type description below) |
| **<numeric_value>** | Inductance L<no> for the specified circuit model. |
| Range [def. unit] | –1H to 1 H [H]. The increment is 1 pH ($10^{-12}$ H). |
| *RST value | 1 nH ($10^{-9}$ H) |
| **SCPI, Command Types** | Device-specific, command or query. In the query form, the <numeric_value> parameter must be omitted. The command returns the inductance value for |

the specified circuit model.

**Example:**

```
*RST; CALC:TRAN:VNET:BAL:EMB:PAR:L1? CSSL
```

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.

## CALCulate<Ch>:TRANsform:VNETworks:BALanced:EMBedding<Log_pt>:PARameters:R<no> STSL | STSC | SLST | SCST | CSSL | LSSC | CSSC | LSSL | SLCS | SCLS | SCCS | SLLS,<numeric_value>

Specifies the resistance values R1, R2, R3 in the different circuit models for balanced port embedding.

**<Ch>** Channel number

**<Log_pt>** Logical port number

**<no>** Number of resistance in circuit model. If unspecified the numeric suffix is set to 1.

Range [def. unit] 1 to 3 [–] The number of resistances depends on the selected circuit model.

**Parameters** Possible circuit models (character data); see circuit models.

*RST value – (see command type description below)

**<numeric_value>** Resistance R<no> for the specified circuit model.

Range [def. unit] –10 MΩ to 10 MΩ [Ω]. The increment is 1 mΩ ($10^{-3}$ Ω).

*RST value 0 Ω for all resistances connected in series with an inductance
10 MΩ for all resistances connected in parallel with a capacitance

**SCPI, Command Types** Device-specific, command or query. In the query form, the <numeric_value> parameter must be omitted. The command returns the resistance value for the specified circuit model.

**Example:**

```
*RST; CALC:TRAN:VNET:BAL:EMB:PAR:R1? CSSL; R2? CSSL, R3? CSSL
```

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.

## CALCulate<Ch>:TRANsform:VNETworks:BALanced:EMBedding<Log_pt>[: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.

**<Ch>** Channel number

**<Log_pt>** Logical port number

**<Boolean>** ON – Embedding active
OFF – Embedding inactive

| | |
|---|---|
| *RST value | OFF [–] |
| **SCPI, Command Types** | Device-specific, command or query |
| **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. |

## CALCulate<Ch>:TRANsform:VNETworks:BALanced:EMBedding<Log_pt>:TNDefinition FIMPort | STSL | STSC | SLST | SCST | CSSL | LSSC | CSSC | LSSL | SLCS | SCLS | SCCS | SLLS

Selects the circuit model for balanced port embedding.

| | |
|---|---|
| **<Ch>** | Channel number |
| **<Log_pt>** | Logical port number |
| **Parameters** | Possible circuit models (character data); see circuit models. |
| *RST value | CSSL |
| **SCPI, Command Types** | Device-specific, command or query. |
| **Example:** | `See CALCulate<Ch>:TRANsform:VNETworks:BALanced:EMbedding[:STATe]` |

## CALCulate<Ch>:TRANsform:VNETworks:SENDed:DEEMbedding<Ph_pt>:PARameters:C<no> CSL | LSC | CSC | SLC | SCL | SCC,<numeric_value>

Specifies the capacitance values C1, C2 in the different circuit models for single ended port deembedding.

| | |
|---|---|
| **<Ch>** | Channel number |
| **<Ph_pt>** | Physical port number |
| **<no>** | Number of capacitance in circuit model. If unspecified the numeric suffix is set to 1. |
| Range [def. unit] | 1 to 2 [–] The number of capacitances depends on the selected circuit model. |
| **Parameters** | Possible circuit models (character data); see circuit models. |
| *RST value | – (see command type description below) |
| **<numeric_value>** | Capacitance C<no> for the specified circuit model. |
| Range [def. unit] | –1mF to 1 mF [F]. The increment is 1 fF ($10^{-15}$ F). |
| *RST value | 1 pF ($10^{-12}$ F) |
| **SCPI, Command Types** | Device-specific, command or query. In the query form, the <numeric_value> parameter must be omitted. The command returns the capacitance value for |

the specified circuit model.

**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.

## CALCulate<Ch>:TRANsform:VNETworks:SENDed:DEEMbedding<Ph_pt>:PARameters:L<no> CSL | LSC | LSL | SLC | SCL | SLL,<numeric_value>

Specifies the inductance values L1, L2, L3 in the different circuit models for single ended port deembedding.

**<Ch>** Channel number.

**<Ph_pt>** Physical port number

**<no>** Number of inductance in circuit model. If unspecified the numeric suffix is set to 1.

Range [def. unit] 1 to 3 [–] The number of inductances depends on the selected circuit model.

**Parameters** Possible circuit models (character data); see circuit models.

*RST value – (see command type description below)

**<numeric_value>** Inductance L<no> for the specified circuit model.

Range [def. unit] –1H to 1 H [H]. The increment is 1 pH ($10^{-12}$ H).

*RST value 1 nH ($10^{-9}$ H)

**SCPI, Command Types** Device-specific, command or query. In the query form, the <numeric_value> parameter must be omitted. The command returns the inductance value for the specified circuit model.

**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.

## CALCulate<Ch>:TRANsform:VNETworks:SENDed:DEEMbedding<Ph_pt>:PARameters:R<no> CSL | LSC | CSC | LSL | SLC | SCL | SCC | SLL,<numeric_value>

Specifies the resistance values R1, R2, R3 in the different circuit models for single ended port deembedding.

**<Ch>** Channel number.

**<Ph_pt>** Physical port number

**<no>** Number of resistance in circuit model. If unspecified the numeric suffix is set to 1.

Range [def. unit] 1 to 3 [–] The number of resistances depends on the selected circuit model.

**Parameters**
Possible circuit models (character data); see circuit models.
*RST value
– (see command type description below)

**<numeric_value>**
Resistance R<no> for the specified circuit model.
Range [def. unit]
–10 MΩ to 10 MΩ [Ω]. The increment is 1 mΩ ($10^{-3}$ Ω).
*RST value
0 Ω for all resistances connected in series with an inductance
10 MΩ for all resistances connected in parallel with a capacitance

**SCPI, Command Types**
Device-specific, command or query. In the query form, the <numeric_value> parameter must be omitted. The command returns the resistance value for the specified circuit model.

**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.

## CALCulate<Ch>:TRANsform:VNETworks:SENDed:DEEMbedding<Ph_pt>[: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.

**<Ch>**
Channel number.

**<Ph_pt>**
Physical port number

**<Boolean>**
ON – Deembedding active
OFF – Deembedding inactive
*RST value
OFF [–]

**SCPI, Command Types**
Device-specific, command or query

**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.

## CALCulate<Ch>:TRANsform:VNETworks:SENDed:DEEMbedding<Ph_pt>:TNDefinition FIMPort | CSL | LSC | CSC | LSL | SLC | SCL | SCC | SLL

Selects the circuit model for single ended port deembedding.

**<Ch>**
Channel number.

**<Ph_pt>**
Physical port number

**Parameters**
Possible circuit models (character data); see circuit models.
*RST value
CSL

**SCPI, Command Types** Device-specific, command or query.

**Example:**
```
See CALCulate<Ch>:TRANsform:VNETworks:SENDed:DEEMbedding[:STATe]
```

## CALCulate<Ch>:TRANsform:VNETworks:SENDed:EMBedding<Ph_pt>:PARameters:C<no> CSL | LSC | CSC | SLC | SCL | SCC,<numeric_value>

Specifies the capacitance values C1, C2 in the different circuit models for single ended port embedding.

**<Ch>** Channel number.

**<Ph_pt>** Physical port number

**<no>** Number of capacitance in circuit model. If unspecified the numeric suffix is set to 1.

Range [def. unit] 1 to 2 [–] The number of capacitances depends on the selected circuit model.

**Parameters** Possible circuit models (character data); see circuit models.

*RST value – (see command type description below)

**<numeric_value>** Capacitance C<no> for the specified circuit model.

Range [def. unit] –1mF to 1 mF [F]. The increment is 1 fF ($10^{-15}$ F).

*RST value 1 pF ($10^{-12}$ F)

**SCPI, Command Types** Device-specific, command or query. In the query form, the <numeric_value> parameter must be omitted. The command returns the capacitance value for the specified circuit model.

**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.

## CALCulate<Ch>:TRANsform:VNETworks:SENDed:EMBedding<Ph_pt>:PARameters:L<no> CSL | LSC | LSL | SLC | SCL | SLL,<numeric_value>

Specifies the inductance values L1, L2, L3 in the different circuit models for single ended port embedding.

**<Ch>** Channel number.

**<Ph_pt>** Physical port number

**<no>** Number of inductance in circuit model. If unspecified the numeric suffix is set to 1.

Range [def. unit] 1 to 3 [–] The number of inductances depends on the selected circuit model.

**Parameters** Possible circuit models (character data); see circuit models.

*RST value – (see command type description below)

**<numeric_value>** Inductance L<no> for the specified circuit model.

| | |
|---|---|
| Range [def. unit] | –1H to 1 H [H]. The increment is 1 pH ($10^{-12}$ H). |
| *RST value | 1 nH ($10^{-9}$ H) |
| **SCPI, Command Types** | Device-specific, command or query. In the query form, the <numeric_value> parameter must be omitted. The command returns the inductance value for the specified circuit model. |
| **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. |

## CALCulate<Ch>:TRANsform:VNETworks:SENDed:EMBedding<Ph_pt>:PARameters:R<no> CSL | LSC | CSC | LSL | SLC | SCL | SCC | SLL,<numeric_value>

Specifies the resistance values R1, R2, R3 in the different circuit models for single ended port embedding.

| | |
|---|---|
| **<Ch>** | Channel number. |
| **<Ph_pt>** | Physical port number |
| **<no>** | Number of resistance in circuit model. If unspecified the numeric suffix is set to 1. |
| Range [def. unit] | 1 to 3 [–] The number of resistances depends on the selected circuit model. |
| **Parameters** | Possible circuit models (character data); see circuit models. |
| *RST value | – (see command type description below) |
| **<numeric_value>** | Resistance R<no> for the specified circuit model. |
| Range [def. unit] | –10 MΩ to 10 MΩ [Ω]. The increment is 1 mΩ ($10^{-3}$ Ω). |
| *RST value | 0 Ω for all resistances connected in series with an inductance<br>10 MΩ for all resistances connected in parallel with a capacitance |
| **SCPI, Command Types** | Device-specific, command or query. In the query form, the <numeric_value> parameter must be omitted. The command returns the resistance value for the specified circuit model. |
| **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. |

## CALCulate<Ch>:TRANsform:VNETworks:SENDed:EMBedding<Ph_pt>[: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.

| | |
|---|---|
| **<Ch>** | Channel number. |

| | |
|---|---|
| **<Ph_pt>** | Physical port number |
| **<Boolean>** | ON – Embedding active<br>OFF – Embedding inactive |
| *RST value | OFF [–] |
| **SCPI, Command Types** | Device-specific, command or query |
| **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. |

### CALCulate<Ch>:TRANsform:VNETworks:SENDed:EMBedding<Ph_pt>:TNDefinition FIMPort | CSL | LSC | CSC | LSL | SLC | SCL | SCC | SLL

Selects the circuit model for single ended port embedding.

| | |
|---|---|
| **<Ch>** | Channel number. |
| **<Ph_pt>** | Physical port number |
| **Parameters** | Possible circuit models (character data); see circuit models. |
| *RST value | CSL |
| **SCPI, Command Types** | Device-specific, command or query. |
| **Example:** | `See CALCulate<Ch>:TRANsform:VNETworks:SENDed:EMBedding[:STATe]` |

## CONFigure...

This subsystem creates and deletes channels and assigns names to channels. The commands are device-specific.

### CONFigure:CHANnel<Ch>:CATalog?

Returns the numbers and names of all channels in the current setup.

| | |
|---|---|
| **<Ch>** | Channel number. This parameter is ignored because the command returns **all** channels. |
| **Response** | String with comma-separated list of channel numbers and names, see example below. If all channels have been deleted the response is an empty string (""). |
| *RST value | – |
| **SCPI, Command Types** | Device-specific, query only. |
| **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.

## CONFigure:CHANnel<Ch>:NAME '<Ch_name>'

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<Ch>:CATalog?` returns a list of all defined channels with their names.

| | |
|---|---|
| **<Ch>** | Number of an existing channel. |
| **'<Ch_name>'** | Channel name, e.g. 'Channel 4'. |
| *RST value | 'Ch1' |
| **SCPI, Command Types** | Device-specific, command or query. |
| **Example:** | `See CONFigure:CHANnel<Ch>:CATalog?` |

## CONFigure:CHANnel<Ch>:NAME:ID? '<Ch_name>'

Queries the channel number (numeric suffix) of a channel with known channel name. A channel name must be assigned before (`CONFigure:CHANnel<Ch>NAME '<Ch_name>'`). `CONFigure:CHANnel<Ch>:CATalog?` returns a list of all defined channels with their names.

| | |
|---|---|
| **<Ch>** | Channel number. This suffix is not relevant and may be omitted (the command returns the actual channel number). |
| **'<Ch_name>'** | Channel name, e.g. 'Channel 4'. |
| *RST value | – |
| **SCPI, Command Types** | Device-specific, command or query. |
| **Example:** | `See CONFigure:CHANnel<Ch>:CATalog?` |

## CONFigure:CHANnel<Ch>:TRACe:REName '<Tr_name>'

Assigns a (new) name to the active trace in channel <Ch>.

| | |
|---|---|
| **<Ch>** | Channel number. |
| **'<Tr_name>'** | Trace name, e.g. 'Trace 4'. |

| | |
|---|---|
| *RST value | – |
| **SCPI, Command Types** | Device-specific, no query. |
| **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'. |

## CONFigure:CHANnel<Ch>[:STATe] <Boolean>

Creates or deletes channel no. <Ch> and selects it as the active channel. `CONFigure:CHANnel<Ch>:NAME` defines the channel name.

*A channel created with `CONFigure:CHANnel<Ch>[:STATe] ON` can be configured but has no trace assigned so that no measurement can be initiated. Use `CALCulate<Ch>:PARameter:SDEFine "<Trace_name>","<Parameter>"` to create a new channel and a new trace.*
*In remote control it is possible to remove all channels. This is in contrast to manual control where at least one channel with one diagram area and one trace must be available.*

| | |
|---|---|
| **<Ch>** | Number of the channel to be created or deleted. |
| **<Boolean>** | ON - Create channel no. <Ch>. If the channel no. <Ch> exists already, it is not modified but selected as the active channel<br>OFF - Delete channel no. <Ch>. |
| *RST value | ON for channel no. 1 (created on *RST), OFF for all other channels. |
| **SCPI, Command Types** | Device-specific, command or query. |
| **Example:** | `See CONFigure:CHANnel<Ch>:CATalog?` |

## CONFigure:TRACe<Trc>:CATalog?

Returns the numbers and names of all traces in the current setup.

| | |
|---|---|
| **<Trc>** | Trace number. This suffix is ignored; the command returns a list of all traces. |
| **Response** | String with comma-separated list of trace numbers and names, see example below. If all traces have been deleted the response is an empty string (""). |
| *RST value | – |
| **SCPI, Command Types** | Device-specific, query only. |

**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: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:ID? 'Ch2Trc2'
```

Query the channel number for trace *Ch2Trc2*. The response is 2.

## CONFigure:TRACe<Trc>:CHANnel:NAME? '<Trc_name>'

Queries the channel name for an existing trace named '<Trc_name>'.

| | |
|---|---|
| **<Trc>** | Trace number. This suffix is ignored; the trace is referenced by its name. |
| **'<Trc_name>'** | Trace name, e.g. 'Ch2Trc2'. |
| *RST value | – (a trace name must be specified whenever the query is used) |
| **SCPI, Command Types** | Device-specific, query only. |
| **Example:** | `See CONFigure:TRACe<Trc>:CATalog?` |

## CONFigure:TRACe<Trc>:CHANnel:NAME:ID? '<Trc_name>'

Queries the channel number (numeric suffix) for an existing trace named '<Trc_name>'.

| | |
|---|---|
| **<Trc>** | Trace number. This suffix is ignored; the trace is referenced by its name. |
| **'<Trc_name>'** | Trace name, e.g. 'Ch2Trc2'. |
| *RST value | – (a trace name must be specified whenever the query is used) |
| **SCPI, Command Types** | Device-specific, query only. |
| **Example:** | `See CONFigure:TRACe<Trc>:CATalog?` |

### CONFigure:TRACe<Trc>:NAME '<Trc_name>'

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<Trc>:CATalog?` returns a list of all traces in the active setup with their names.

| | |
|---|---|
| **<Trc>** | Number of an existing trace. |
| **'<Trc_name>'** | Trace name, e.g. 'Ch2Trc2'. |
| *RST value | 'Trc1' |
| **SCPI, Command Types** | Device-specific, command or query. |
| **Example:** | `See CONFigure:TRACe<Trc>:CATalog?` |

### CONFigure:TRACe<Trc>:NAME:ID? '<Trc_name>'

Queries the trace number (numeric suffix) of a trace with known trace name. `CONFigure:TRACe<Trc>:CATalog?` returns a list of all traces in the active setup with their names.

| | |
|---|---|
| **<Trc>** | Trace number. This suffix is not relevant and may be omitted (the command returns the actual trace number). |
| **'<Trc_name>'** | Trace name, e.g. 'Ch2Trc2'. |
| *RST value | – |
| **SCPI, Command Types** | Device-specific, command or query. |
| **Example:** | `See CONFigure:TRACe<Trc>:CATalog?` |

### CONFigure:TRACe<Trc>:REName '<Old_Tr_Name>', '<New_Tr_Name>'

Assigns a new name to a trace. The trace does not have to be the active trace.

| | |
|---|---|
| **<Trc>** | Trace number. This suffix is ignored; the trace is identified via its <Old_Tr_Name> |
| **'<Old_Tr_Name>'**<br>**'<New_Tr_Name>'** | Old trace name, e.g. 'Trace 1'<br>New trace name, e.g. 'Trace 2' |
| *RST value | – |
| **SCPI, Command Types** | Device-specific, no query. |
| **Example:** | `See CONFigure:CHANnel<Ch>:TRACe:REName` |

## CONTrol...

This subsystem provides device-specific commands to control hardware settings related to the rear panel connectors.

## CONTrol:AUXiliary:C[:DATA] <numeric_value>

Sets or queries a channel-dependent four-bit decimal value to control four independent output signals at the USER CONTROL connector (applied to pins 8, 9, 10, 11). The output signals are 3.3 V TTL signals which can be used to differentiate between up to 16 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 of the USER CONTROL connector.
- The value defined with `CONTrol:AUXiliary:C[:DATA]` is assigned to the **active** channel (`INSTrument:NSELect <Ch>`).
- The signals at the USER CONTROL 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 CONTROL 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.

TIPS A simple application consists of selecting the channel numbers as parameters for `CONTrol:AUXiliary:C[:DATA]` and monitor the activity of up to 16 different channels at the USER CONTROL connector; see example below. You can also use the USER CONTROL output signals as channel-dependent trigger signals for external devices. Use OUTPut<Ch>:UPORt[:VALue] to transfer the four bit value for an arbitrary channel <Ch> in binary representation.

| | |
|---|---|
| **<numeric_value>** | Decimal value . |
| Range [def. unit] | 0 to 15 [–]<br>The transferred values correspond to the following states of the USER CONTROL connector:<br>0 – no signal at any of the four pins 8, 9, 10, 11<br>1 – output signal at pin 8<br>2 – output signal at pin 9<br>3 – output signal at pin 8 and 9<br>...<br>15 – output signal at pin 8, 9, 10 and 11 |
| *RST value | 0 (no signal) |
| **SCPI, Command Types** | Confirmed, command or query |
| **Example:** | `*RST; CONT:AUX:C 1`<br>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.<br>`CONF:CHAN2:STAT ON; CONT:AUX:C 2`<br>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.

# DIAGnostic:SERVice...

This subsystem provides 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 should be used by a R&S service representative only. Refer to the service manual for more information.

## DIAGnostic:ALC:SETTings[:STATe] ON | OFF

Enables or disables the Automatic Level Control for all channels.

| | |
|---|---|
| **<Ch>** | Channel number. |
| **<Boolean>** | ON \| OFF - ALC on or off. |
| *RST value | OFF |
| **SCPI, Command Types** | Device-specific, command or query. |
| **Example:** | `DIAG:ALC:SETT OFF` |
| | Disable the *ALC* function. |

## DIAGnostic:SERVice:FUNCtion <numeric_value>,...

## DIAGnostic:SERVice:SFUNction '<string>',...

Activates a service function (for internal use only).

## 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]`.

| | |
|---|---|
| **<Boolean>** | ON \| OFF - Switches the power on or off. |
| *RST value | ON |
| **SCPI, Command Types** | Device-specific, command or query |
| **Example:** | `DIAG:SERV:RFP OFF` |

Turn off the RF source power.

# DISPlay...

This subsystem controls the selection and presentation of graphical and trace information on the screen.

***Note:*** *Traces are generally identified by a string parameter defining the trace name (e.g.* `CALCulate<Ch>:PARameter:SELect <Trace_Name>`*). In the DISPlay... subsystem, traces are assigned to diagram areas (*`DISPlay:WINDow<Wnd>:TRACe<WndTr>:FEED <Trace_Name>`*). While this assignment is valid, the trace is identified by the numeric suffix* `<WndTr>` *and the trace name is not needed.*

## DISPlay:ANNotation:FREQuency[:STATe] <Boolean>

Shows or hides all frequency stimulus values in the diagrams.

| | |
|---|---|
| **<Boolean>** | ON \| OFF - Show or hide stimulus values. |
| *RST value | ON |
| **SCPI, Command Types** | Confirmed, command or query |
| **Example:** | `*RST; DISP:ANN:FREQ OFF` |
| | Create diagram area no. 1 (with default trace) and hide the frequency stimulus values. |

## DISPlay:CMAP<Element>: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<Element>:RGB <red>, <green>, <blue>,....`

| | |
|---|---|
| **<Element>** | Numeric suffix, not used in this command. Implemented for compatibility with DISPlay:CMAP<Element>:RGB. |
| **<Boolean>** | ON – All markers have the same color, to be defined via `DISPlay:CMAP6:RGB <red>, <green>, <blue>`. The marker color is independent of the trace colors.<br>OFF - Each marker has the color of the associated trace. |
| *RST value | *RST does not affect the color settings; see also description of the Preset command. |
| **SCPI, Command Types** | Device-specific, command or query |
| **Example:** | `See DISPlay:CMAP<Element>:RGB.` |

## DISPlay:CMAP<Element>:RGB <red>, <green>, <blue> [,<trace_style>, <trace_width>]

Defines the color of all display elements based on the Red/Green/Blue color model.

**<Element>** Number of the display element. The display elements corresponding to the numbers 1 to 28 are listed below.

**<red>**
**<green>**
**<blue>**
Red, green, and blue content of the defined color.

Range [def. unit] 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) [–]

**<trace_style>** Optional trace style, only for traces (<Element> > 12): One of the string parameters SOLid | DASHed | DOTTed | DDOTted | DDDotted.

**<trace_width>** Optional trace width, only for traces (<Element> > 12).

Range [def. unit] 1 to 20 [–]

*RST values *RST does not affect the color settings; see also description of the Preset command.

**SCPI, Command Types** Confirmed (with device-specific numeric suffix and parameters), command or query. The query returns three values between 0 and one, separated by commas, corresponding to the red, green, and blue color content.

**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 *Mkr 1*.

```
CALC:PAR:SDEF 'TRC2', 'S11'; DISP:WIND:TRAC2:FEED 'TRC2'
```

Create a new 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 *Mkr 2* 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: *Mkr 1* turns red, *Mkr 2* turns green.

The numeric suffixes <*Element*> denote the following display elements:

| <Element> | Display Element |
|---|---|
| 1 | Background |
| 2 | Text |

| | |
|---|---|
| 3 | Selected Text |
| 4 | Grid |
| 5 | Reference Line |
| 6 | Same Color for all Markers |
| 7 | Horizontal Line / 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 | Trace 1 (see also `DISPlay:CMAP<Element>:TRACe:RGB`) |
| 14 | Trace 2 |
| 15 | Trace 3 |
| 16 | Trace 4 |
| 17 | Trace 5 |
| 18 | Trace 6 |
| 19 | Trace 7 |
| 20 | Trace 8 |
| 21 | Trace 9 |

| | |
|---|---|
| 22 | Trace 10 |
| 23 | Trace 11 |
| 24 | Trace 12 |
| 25 | Trace 13 |
| 26 | Trace 14 |
| 27 | Trace 15 |
| 28 | Trace 16 |

## DISPlay:CMAP<Element>:TRACe:COLor[:STATe] <Boolean>

Defines the trace color schemes in different diagram areas.

**<Element>** Numeric suffix, not used in this command. Implemented for compatibility with DISPlay:CMAP<Element>:RGB.

**<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.

*RST value *RST does not affect the color settings; see also description of the Preset command.

**SCPI, Command Types** Device-specific, command or query

**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.

## DISPlay:CMAP<Element>:TRACe:RGB '<trace_name>, <red>, <green>, <blue> [,<trace_style>, <trace_width>]

Defines the color of a trace referenced by its name, based on the Red/Green/Blue color model. Use the generalized command `DISPlay:CMAP<Element>:RGB` to define the color of other display elements.

| | |
|---|---|
| **<Element>** | Numeric suffix, not used in this command. Implemented for compatibility with DISPlay:CMAP<Element>:RGB. |
| **'<trace_name>'** | Trace name, string parameter |
| **<red>**<br>**<green>**<br>**<blue>** | Red, green, and blue content of the defined color. |
| Range [def. unit] | 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) [–] |
| **<trace_style>** | Optional trace style, only for traces (<Element> > 12): One of the string parameters SOLid \| DASHed \| DOTTed \| DDOTted \| DDDotted. |
| **<trace_width>** | Optional trace width, only for traces (<Element> > 12). |
| Range [def. unit] | 1 to 20 [–] |
| *RST values | *RST does not affect the color settings; see also description of the Preset command. |
| **SCPI, Command Types** | Device-specific, command or query. The query returns three values between 0 and one, separated by commas, corresponding to the red, green, and blue color content. |
| **Example:** | `*RST; DISP:CMAP:TRAC:RGB 'Trc1', 1, 0, 0` |
| | Color the default trace 'Trc1' red.. |
| | `See also DISPlay:CMAP<Element>:RGB` |

## DISPlay:MENU:KEY:EXECute '<menu_key>'

Executes the function of a key with a specified name and switches to the local screen.

You can use this command to execute part of your measurement task manually; see *Combining Manual and Remote Control*. Menu keys that initiate events are executed immediately, because no additional input is necessary. However, `DISPlay:MENU:KEY:EXECute` can be used as well for manual entries in numeric entry bars, dialogs or wizards.

| | |
|---|---|
| **'<menu_key>'** | Name of the key as shown in the analyzer's softkey bar (case-sensitive string variable, may contain blanks but no dots). If the name is not unique, the complete menu path can be specified, e.g. 'Trace:Format:Phase'. Menus, submenus and menu function must be separated by colons. |
| *RST value | – |

**SCPI, Command Types** Device-specific, no query

**Example:**

```
*RST; DISP:MENU:KEY:EXEC 'S11'; EXEC 'Trace:Format:Phase'
```

Assign the S-parameter $S_{11}$ to the default trace, open the local analyzer screen, then display the phase of the measured quantity. The commands are executed immediately without any manual entry.

```
DISP:MENU:KEY:EXEC 'Start'
```

Open the numeric input bar for the start frequency of the sweep. The frequency can be entered manually.

```
DISP:MENU:KEY:EXEC 'About Nwa'
```

Display information about your network analyzer and the firmware version. The info box is closed when you click the analyzer screen or if you send another command.

## DISPlay:MENU:KEY:SELect '<menu_key>'

Activates the menu or submenu of the specified key.

**'<menu_key>'** Name of the key as shown in the analyzer's softkey bar (case-sensitive string variable).

*RST value –

**SCPI, Command Types** Device-specific, no query

**Example:**

```
*RST; DISP:MENU:KEY:SEL 'S11'
```

Open the *Trace – Meas* menu in order to select a measured quantity.

## DISPlay[:WINDow<Wnd>]:MAXimize <Boolean>

Maximizes all diagram areas in the active setup or restores the previous display configuration.

**<Wnd>** Number of the diagram area to become the active diagram area.
`DISPlay:WINDow<Wnd>:MAXimize` acts on all diagrams of the current setup, however, the diagram no. <Wnd> is displayed on top of the others.

**<Boolean>** ON | OFF - Maximize all diagram areas or restore the previous display configuration.

*RST value OFF

**SCPI, Command Types** Device-specific, command or query (returns whether or not the diagrams are maximized).

**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.

## DISPlay[:WINDow<Wnd>]:STATe <Boolean>

Creates or deletes a diagram area, identified by its area number <Wnd>.

| | |
|---|---|
| **<Wnd>** | Number of the diagram area to be created or deleted. |
| **<Boolean>** | ON \| OFF - Creates or deletes diagram area no. <Wnd>. |
| *RST value | – |
| **SCPI, Command Types** | Device-specific, command or query (returns whether or not a particular diagram area exists). |
| **Example:** | `CALC4:PAR:SDEF 'Ch4Tr1', 'S11'` |
| | Create channel 4 and a trace named *Ch4Tr1* to measure the input reflection coefficient S11. |
| | `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. |

## DISPlay[:WINDow<Wnd>]:TITLe:DATA '<string>'

Defines a title for diagram area <Wnd>.

| | |
|---|---|
| **<Wnd>** | Number of the diagram area. |
| **'<string>'** | 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. |
| *RST value | '' |
| **SCPI, Command Types** | Device-specific, command or query |
| **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. |

## DISPlay[:WINDow<Wnd>]:TITLe[:STATe] <Boolean>

Displays or hides the title for area number <Wnd>, defined by means of `DISPlay:WINDow<Wnd>:TITLe:DATA`.

| | |
|---|---|
| **<Wnd>** | Number of the diagram area. |
| **<Boolean>** | ON \| OFF - Displays or hides the title. |
| *RST value | ON |
| **SCPI, Command Types** | Device-specific, command or query |
| **Example:** | `See DISPlay:WINDow<Wnd>:TITLe:DATA.` |

## 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 <Trace_Name>` and expressed by the <WndTr> suffix. The trace itself is not deleted; this must be done via `CALCulate<Ch>:PARameter:DELete <Trace_Name>`.

| | |
|---|---|
| **<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>. |
| *RST value | – |
| **SCPI, Command Types** | Device-specific, no query. |
| **Example:** | `CALC4:PAR:SDEF 'Ch4Tr1', 'S11'` |
| | Create channel 4 and a trace named *Ch4Tr1* to measure the input reflection coefficient S11. |
| | `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.* |

## DISPlay[:WINDow<Wnd>]:TRACe:EFEed '<trace_name>'

Assigns an existing trace (`CALCulate<Ch>:PARameter:SDEFine <Trace_Name>`) 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).

*You can open the trace manager (`DISPlay:MENU:KEY:EXECute 'Trace Manager'`) to obtain an overview of all channels and traces, including the traces that are not displayed.*

**<Wnd>** Number of an existing diagram area (defined by means of `DISPlay:WINDow<Wnd>:STATe ON`).

**'<trace_name>'** String parameter for the trace name, e.g. 'Trc4'.

*RST value –

**SCPI, Command Types** Device-specific, no query.

**Example:**

```
CALC4:PAR:SDEF 'Ch4Tr1', 'S11'
```

Create channel 4 and a trace named *Ch4Tr1* to measure the input reflection coefficient S11.

```
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.

## DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:FEED '<trace_name>'

Assigns an existing trace (`CALCulate<Ch>:PARameter:SDEFine <Trace_Name>`) 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.

*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 (`DISPlay:MENU:KEY:EXECute 'Trace Manager'`) to obtain an overview of all channels and traces, including the traces that are not displayed.*

**<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>.

**'<trace_name>'** String parameter for the trace name, e.g. 'Trc4'.

*RST value –

**SCPI, Command Types** Device-specific, no query.

**Example:**

```
CALC4:PAR:SDEF 'Ch4Tr1', 'S11'
```

Create channel 4 and a trace named *Ch4Tr1* to measure the input reflection coefficient S11.

```
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.

## DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:X:OFFSet <numeric_value>

Shifts the trace <WndTr> in horizontal direction, leaving the positions of all markers unchanged.

| | |
|---|---|
| **<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`. |
| **<numeric_value>** | Stimulus offset value. |
| Range [def. unit] | –1000 GHz to +1000 GHz [Hz, for frequency sweeps]. The range and unit depends on the sweep type (`[SENSe<Ch>:]SWEep:TYPE`). |
| *RST value | 0. |
| **SCPI, Command Types** | Device-specific, with query. |
| **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. |

## DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:Y:OFFSet <Magnitude>[,<Phase>, <Real>, <Imaginary>]

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:

$$M_{new} = M_{old} \cdot 10^{<\text{Magnitude}>/20\,\text{dB}} \cdot a \cdot e^{j \cdot <\text{Phase}>/180^\circ} + <\text{Real}> + j <\text{Imag}>$$

| | |
|---|---|
| **<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`. |
| **<Magnitude>** | Multiplied magnitude factor |
| Range [def. unit] | –300 dB to + 300 dB [dB] |
| *RST value | 0 dB |
| **<Phase>** | Multiplied phase factor, optional for setting command but returned by query |
| Range [def. unit] | $-3.4*10^{38}$ deg to $+3.4*10^{38}$ deg [deg] |
| *RST value | 0 deg |
| **<Real>, <Imaginary>** | Real and imaginary part of added complex constant, optional for setting command but returned by query |

| | |
|---|---|
| Range [def. unit] | $-3.4*10^{38}$ to $+3.4*10^{38}$ [–] |
| *RST value | 0. |
| **SCPI, Command Types** | Device-specific, with query. |
| **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.* |

## DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:Y[:SCALe]:AUTO ONCE[, '<trace_name>']

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 <trace_name>.

| | |
|---|---|
| **<Wnd>** | Number of an existing diagram area (defined by means of `DISPlay:WINDow<Wnd>:STATe ON`). This suffix is ignored if the optional <trace_name> parameter is used. |
| **<WndTr>** | Existing trace number, assigned by means of `DISPlay:WINDow<Wnd>:TRACe<WndTr>:FEED`. This suffix is ignored if the optional <trace_name> parameter is used. |
| **ONCE** | Activate the autoscale function. |
| **'<trace_name>'** | 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). |
| *RST value | – |
| **SCPI, Command Types** | Confirmed, no query. |
| **Example:** | `*RST; DISP:WIND:TRAC:Y:PDIV?; RLEV?` |
| | Query the value between two grid graticules and the reference value for the default trace. The response is *10;0.* |
| | `DISP:WIND:TRAC:Y:AUTO ONCE; PDIV?; RLEV?`<br>**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. |

## DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:Y[:SCALe]:BOTTom <lower_value>[, '<trace_name>']

Sets the lower (minimum) edge of the diagram area <Wnd>.

| | |
|---|---|
| **<Wnd>** | Number of an existing diagram area (defined by means of `DISPlay:WINDow<Wnd>:STATe ON`). This suffix is ignored if the optional |

<trace_name> parameter is used.

**<WndTr>** Existing trace number, assigned by means of `DISPlay:WINDow<Wnd>:TRACe<WndTr>:FEED`. This suffix is ignored if the optional <trace_name> parameter is used.

**<lower_value>** Value and unit for the lower diagram edge.

Range [def. unit] Range and unit depend on the measured quantity, see *Units for DISPlay... Commands*.

*RST value Depending on the measured quantity. The default lower edge for a *dB Mag* diagram is –80 dB.

**'<trace_name>'** 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).

*RST value –

**SCPI, Command Types** Confirmed, with query.

**Example:**

```
CALC4:PAR:SDEF 'Ch4Tr1', 'S11'
```

Create channel 4 and a trace named *Ch4Tr1* to measure the input reflection coefficient S11.

```
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.

## DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:Y[:SCALe]:PDIVision <numeric_value>[, '<trace_name>']

Sets the value between two grid graticules (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.

**<Wnd>** Number of an existing diagram area (defined by means of `DISPlay:WINDow<Wnd>:STATe ON`). This suffix is ignored if the optional <trace_name> parameter is used.

**<WndTr>** Existing trace number, assigned by means of `DISPlay:WINDow<Wnd>:TRACe<WndTr>:FEED`. This suffix is ignored if the optional <trace_name> parameter is used.

**<numeric_value>** Value and unit for the vertical diagram divisions.

Range [def. unit] Range and unit depend on the measured quantity, see *Units for DISPlay... Commands*.

*RST value Depending on the measured quantity. The default reference level for an S-parameter displayed in a *dB Mag* diagram is 10 dB.

**'<trace_name>'** 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).

*RST value –

**SCPI, Command Types** Confirmed, with query.

**Example:**

```
CALC4:PAR:SDEF 'Ch4Tr1', 'S11'
```

Create channel 4 and a trace named *Ch4Tr1* to measure the input reflection coefficient S11.

```
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.

## DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:Y[:SCALe]:RLEVel <numeric_value>[, '<trace_name>']

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 <trace_name>.

**<Wnd>** Number of an existing diagram area (defined by means of `DISPlay:WINDow<Wnd>:STATe ON`). This suffix is ignored if the optional <trace_name> parameter is used.

**<WndTr>** Existing trace number, assigned by means of `DISPlay:WINDow<Wnd>:TRACe<WndTr>:FEED`. This suffix is ignored if the optional <trace_name> parameter is used.

**<numeric_value>** Value and unit for the reference level (or reference value, if the trace does not show a level).

Range [def. unit] Range and unit depend on the measured quantity, see *Units for DISPlay... Commands*.

*RST value Depending on the measured quantity. The default reference level for an S-parameter displayed in a *dB Mag* diagram is 0 dB.

**'<trace_name>'** 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).

*RST value –

**SCPI, Command Types** Confirmed, with query.

**Example:**

```
CALC4:PAR:SDEF 'Ch4Tr1', 'S11'
```

Create channel 4 and a trace named *Ch4Tr1* to measure the input reflection coefficient S11.

```
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.

## DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:Y[:SCALe]:RPOSition <numeric_value>[, '<trace_name>']

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.

**<Wnd>** Number of an existing diagram area (defined by means of `DISPlay:WINDow<Wnd>:STATe ON`). This suffix is ignored if the optional <trace_name> parameter is used.

**<WndTr>** Existing trace number, assigned by means of `DISPlay:WINDow<Wnd>:TRACe<WndTr>:FEED`. This suffix is ignored if the optional <trace_name> parameter is used.

**<numeric_value>** 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 [def. unit] 0 to 100 [PCT].
*RST value 80 PCT.

**'<trace_name>'** 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).

*RST value –

**SCPI, Command Types** Confirmed, with query.

**Example:**

```
CALC4:PAR:SDEF 'Ch4Tr1', 'S11'
```

Create channel 4 and a trace named *Ch4Tr1* to measure the input reflection coefficient S11.

```
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.

### DISPlay[:WINDow<Wnd>]:TRACe<WndTr>:Y[:SCALe]:TOP <upper_value>[, '<trace_name>']

Sets the upper (maximum) edge of the diagram area <Wnd>.

| | |
|---|---|
| **<Wnd>** | Number of an existing diagram area (defined by means of `DISPlay:WINDow<Wnd>:STATe ON`). This suffix is ignored if the optional <trace_name> parameter is used. |
| **<WndTr>** | Existing trace number, assigned by means of `DISPlay:WINDow<Wnd>:TRACe<WndTr>:FEED`. This suffix is ignored if the optional <trace_name> parameter is used. |
| **<upper_value>** | Value and unit for the upper diagram edge. |
| Range [def. unit] | Range and unit depend on the measured quantity, see *Units for DISPlay... Commands*. |
| *RST value | Depending on the measured quantity. The default upper edge for a *dB Mag* diagram is +20 dB. |
| **'<trace_name>'** | 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). |
| *RST value | – |
| **SCPI, Command Types** | Confirmed, with query. |
| **Example:** | `See DISPlay:WINDow<Wnd>:TRACe<WndTr>:Y[:SCALe]:BOTTom` |

## 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. This requires the entry of a numeric value and a physical unit, depending on the parameter type displayed. The following table lists the physical units accepted by the analyzer.

| | |
|---|---|
| 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` |

# FORMat...

This subsystem sets a data format for transferring numeric and array information from and to the analyzer.

## FORMat:BORDer NORMal | SWAPped

Controls whether binary data is transferred in normal or swapped byte order.

| | |
|---|---|
| **Parameters** | SWAPped – The least significant bit is transferred first (little endian)<br>NORMal – The most significant bit is transferred first (big endian) |
| *RST value | SWAPped (if the *GPIB Language* is set to *PNA* or *HP xxxx,* then the order is NORMal) |
| **SCPI, Command Types** | Confirmed, command or query |
| **Example:** | `FORM:BORD NORM` |
| | Change the bit order to normal mode. |

## FORMat[:DATA] ASCii | REAL [,<length>]

Selects the format for numeric data transferred to and from the analyzer.

*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.*

| | |
|---|---|
| **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 block data format. |

*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 value | ASCII. The default length of REAL data is 32 bits (single precision). |
| **SCPI, Command Types** | Confirmed, command or query |
| **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_of_bytes> bytes binary data received*.

### FORMat:DEXPort:SOURce FDATa | SDATa | MDATa

Defines the format for traces retrieved with the ZVR-compatible command `TRACe[:DATA][:RESPonse][:ALL]?`

This command is not relevant for results read with the `CALCulate:DATA...` commands.

| | |
|---|---|
| **Parameters** | See list of parameters below. |
| Range [def. unit] | Depending on the measured parameter and format. The unit is the default unit of the measured parameter; see `CALCulate<Ch>:PARameter:SDEFine`. |
| *RST value | SDATa |
| **SCPI, Command Types** | Device-specific, command or query. |
| **Example:** | `See TRACe[:DATA][:RESPonse][:ALL]?` |

The following parameters are related to trace data (see also *Data Flow* diagram):

| | |
|---|---|
| `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. |

## HCOPy...

The HCOPy system controls the output of screen information to an external device. Part of the functionality of this system is included in the *File* menu.

## HCOPy:DESTination <string> | 'MMEM'

Selects a printer name or file as destination for the screen output.

| | |
|---|---|
| **<string> \|** | Printer name, string variable. One of the printers accessible from your PC. |
| **'MMEM'** | Print to file. The file name is defined via `MMEMory:NAME`. The command `HCOPy:DEVice:LANGuage WMF \| EWMF \| BMP \| PNG` selects the file format. |
| *RST value | – <br> ***Note:*** **RST does not overwrite the printer destination.* |
| **SCPI, Command Types** | Confirmed, command or query. |

**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.

## HCOPy:DEVice:LANGuage EMF | WMF | EWMF | BMP | JPG | PNG

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.

| | |
|---|---|
| **EMF, WMF, EWMF** | Windows enhanced metafile format (vector-based, file extension *.emf). The parameter WMF is equivalent to EMF; a query returns EMF. |
| **BMP** | Windows bitmap |
| **JPG** | JPEG bitmap |
| **PNG** | Portable network graphics format |
| *RST value | BMP |
| **SCPI, Command Types** | Confirmed, command or query. |

**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]

Immediately initializes the print according to the current HCOPy setup parameters.

| | |
|---|---|
| *RST value | – |
| **SCPI, Command Types** | Confirmed, no query. |

**Example:**

```
HCOP:DEST '<Printer_name>'
```

Select the printer for the output of screen data.

```
HCOP
```

```
Start printing.
```

## HCOPy:ITEM:ALL

Selects the complete screen contents to be printed, including the logo (HCOPy:ITEM:LOGO), time (HCOPy:ITEM:TIME), and the marker list (HCOPy:ITEM:MLISt).

*RST value: After a reset, the complete information is printed; see commands for logo, time, marker list and settings.

**SCPI, Command Types:** Confirmed, no query.

**Example:**

```
HCOP:ITEM:ALL
```

Select the complete information to be printed.

```
HCOP
```

```
Start printing.
```

## 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 NWA program directory and can be replaced by another logo.

**<Boolean>:** ON | OFF - Logo is included or excluded.

*RST value: OFF

**SCPI, Command Types:** Device-specific, command or query

**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).

**<Boolean>:** ON | OFF - Marker list is included or excluded.

*RST value: OFF

**SCPI, Command Types:** Device-specific, command or query

**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.

| | |
|---|---|
| **<Boolean>** | ON \| OFF - Date and time is included or excluded. |
| *RST value | OFF |
| **SCPI, Command Types** | Device-specific, command or query |
| **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:MARGin:BOTTom <numeric_value>

Defines the distance between the bottom of the page and the bottom of the printed information.

| | |
|---|---|
| **<numeric_value>** | Lower margin. |
| Range [def. unit] | 0.01 mm to 10000 mm [mm] |
| *RST value | 0.01 mm |
| **SCPI, Command Types** | Device-specific, command or query |
| **Example:** | `HCOP:PAGE:MARG:BOTT 10; TOP 10` |
| | Set an upper and a lower margin of 1 cm. |

## HCOPy:PAGE:MARGin:LEFT <numeric_value>

Defines the distance between the left edge of the page and the left edge of the printed information.

| | |
|---|---|
| **<numeric_value>** | Left margin. |
| Range [def. unit] | 0.01 mm to 10000 mm [mm] |
| *RST value | 0.01 mm |
| **SCPI, Command Types** | Device-specific, command or query |
| **Example:** | `HCOP:PAGE:MARG:LEFT 10; RIGHT 10` |
| | Set an left and a right margin of 1 cm. |

## HCOPy:PAGE:MARGin:RIGHt <numeric_value>

Defines the distance between the right edge of the page and the right edge of the printed information.

| | |
|---|---|
| **<numeric_value>** | Right margin. |
| Range [def. unit] | 0.01 mm to 10000 mm [mm] |

| | |
|---|---|
| *RST value | 0.01 mm |
| **SCPI, Command Types** | Device-specific, command or query |
| **Example:** | `HCOP:PAGE:MARG:LEFT 10; RIGHT 10` |
| | Set an left and a right margin of 1 cm. |

## HCOPy:PAGE:MARGin:TOP <numeric_value>

Defines the distance between the top of the page and the top of the printed information.

| | |
|---|---|
| **<numeric_value>** | Upper margin. |
| Range [def. unit] | 0.01 mm to 10000 mm [mm] |
| *RST value | 0.01 mm |
| **SCPI, Command Types** | Device-specific, command or query |
| **Example:** | `HCOP:PAGE:MARG:BOTT 10; TOP 10` |
| | Set an upper and a lower margin of 1 cm. |

## HCOPy:PAGE:ORIentation PORTrait | LANDscape

Defines the orientation of the printed page. Switching between LANDscape and PORTrait rotates the hardcopy result by 90 degrees. No other settings are changed.

| | |
|---|---|
| **PORTrait \|** | Short edge of the paper is the top of the page. |
| **LANDscape** | Long edge of the paper is the top of the page. |
| *RST value | PORTrait |
| **SCPI, Command Types** | Confirmed, command or query |
| **Example:** | `HCOP:PAGE:ORI LAND; HCOP` |
| | Select landscape page orientation and start printing. |

## HCOPy:PAGE:WINDow ACTive | ALL | SINGle

Defines the number of diagram areas per printed page.

| | |
|---|---|
| **ACTive** | Print only active diagram area. |
| **ALL \|** | All diagram areas are printed on one page. |
| **SINGle** | One diagram area per page. |
| *RST value | ALL |
| **SCPI, Command Types** | Confirmed, command or query |
| **Example:** | `HCOP:PAGE:WIND SING; HCOP` |
| | Select one diagram area per page and start printing. |

# INITiate<Ch>...

This subsystem control the initiation of the trigger system and defines the scope of the triggered measurement.

## INITiate<Ch>:CONTinuous <Boolean>

Qualifies whether the analyzer measures in single sweep or in continuous sweep mode.

| | |
|---|---|
| **<Ch>** | Channel number. The channel is ignored because the command affects all channels. |
| **<Boolean>** | ON - The analyzer measures continuously, repeating the current sweep.<br>OFF - The measurement is stopped after the number of sweeps defined via `[SENSe<Ch>:]SWEep:COUNt`. INITiate<Ch>[:IMMediate] initiates a new measurement cycle. |
| *RST value | ON |
| **SCPI, Command Types** | Confirmed, command or query |
| **Example:** | `*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 only. |
| | `CALC2:PAR:SDEF 'TRC2','S11'; INIT2` |
| | Create channel no. 2 with a new trace and start a single sweep in channel no. 2. |

## INITiate<Ch>[:IMMediate]

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 *Sweep History)* can be read using CALCulate<Ch>:DATA:NSWeep? SDATa, <history_count>.

*In contrast to all other commands of the analyzer, `INITiate<Ch>[:IMMediate]` has been implemented for overlapped execution; see Command Synchronization.*

| | |
|---|---|
| **<Ch>** | Channel number. If the channel number does not exist the analyzer returns an error message. If `INITiate<Ch>[:IMMediate]:SCOPe ALL` is active, this suffix is ignored. |
| *RST value | – |
| **SCPI, Command Types** | Confirmed, no query |
| **Example:** | `See INITiate<Ch>:CONTinuous` |

### INITiate<Ch>[:IMMediate]:SCOPe ALL | SINGle

Selects the scope of the single sweep sequence. The setting is applied in single sweep mode only (INITiate<Ch>:CONTinuous OFF).

| | |
|---|---|
| **<Ch>** | Channel number. The channel is ignored because the command affects all channels. |
| **ALL** | INITiate<Ch>[:IMMediate] starts a single sweep in all channels. |
| **SINGle** | INITiate<Ch>[:IMMediate] starts a single sweep in the referenced channel <Ch> only. |
| *RST value | ALL |
| **SCPI, Command Types** | Device-specific, command or query |
| **Example:** | `See INITiate<Ch>:CONTinuous` |

## INPut...

This subsystem controls the characteristics of the analyzer's input ports.

### INPut<port_no>:ATTenuation <numeric_value>

Sets the attenuation for the received waves. The generated wave is attenuated via `OUTPut<port_no>:ATTenuation`.

***Note:*** *`INPut<port_no>: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.*

| | |
|---|---|
| **<port_no>** | Test port number of the analyzer, 1 to 2, 3 or 4, depending on the analyzer model. If unspecified the numeric suffix is set to 1. |
| **<numeric_value>** | Attenuation factor for the received wave. |
| Range [def. unit] | Depending on the analyzer/attenuator model, e.g. 0 dB to 75 dB for ZVA8/ZVA8-B31/32/B33/B34 [dB]. UP and DOWN increment/decrement the attenuation in 5 or 10 dB steps. The analyzer rounds any entered value below the maximum attenuation to the closest step. |
| *RST value | 0 dB |
| **SCPI, Command Types** | Confirmed, no query (query can not return channel-specific settings, would be misleading). |
| **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...

This subsystem identifies and selects particular resources (SCPI: logical instruments) of the analyzer.

## INSTrument:NSELect <Ch>

Selects a channel <Ch> as the active channel.

| | |
|---|---|
| **<Ch>** | Number of the channel to be activated. The channel must be created before using `CONFigure:CHANnel<Ch>:STATe ON`. |
| Range [def. unit] | 1, 2, ... [–] |
| *RST value | 1 |
| **SCPI, Command Types** | Confirmed, command or query. If all channels have been deleted `CONFigure:CHANnel<Ch>:STATe OFF` the query returns 0. |
| **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 of the analyzer.

| | |
|---|---|
| **Response** | Number of ports (integer number). |
| Range [def. unit] | 1, 2, 3 or 4, depending on the analyzer model. |
| *RST value | – |
| **SCPI, Command Types** | Device-specific, query only. |
| **Example:** | `INST:PORT:COUN?` |
| | `Return the number of ports of your analyzer.` |

## INSTrument[:SELect] CHANNEL1 | CHANNEL2 | CHANNEL3 | CHANNEL4

Selects a channel <Ch> as active channel.

To select a channel number > 4 use the generalized command `INSTrument:NSELect <Ch>`.

| | |
|---|---|
| **Parameters** | Number of the channel to be activated. The channel must be created before using `CONFigure:CHANnel<Ch>:STATe ON`. |
| Range [def. unit] | CHANNEL1, CHANNEL2, CHANNEL3, CHANNEL4 [–] |
| *RST value | CHANNEL1 |
| **SCPI, Command** | Confirmed, command or query. If all channels have been deleted (`CONFigure:CHANnel<Ch>:STATe OFF`) the query returns CHANNEL0. |

**Types**

**Example:**

```
CONF:CHAN2:STAT ON; INST CHANNEL2
```

Create channel no. 2 and select it as the active channel.

## MEMory...

The MEMory system controls the loaded setups of the analyzer.

The MEMory... command don't affect any stored files. Use the MEMory... commands to store and load data and to manage files stored on a mass storage device.

### MEMory:CATalog?

Returns the names of all loaded setups.

**Response** Comma-separated list of all setups.

*RST value 'Set1' (the default setup is created after each *RST).

**SCPI, Command Types** Confirmed (with device-specific response), query only.

**Example:**

```
*RST; MEM:DEF 'SETUP_2'
```

Create a setup named 'Setup_2' and make it the active setup.

```
MEM:CAT?
```

Query all setups. The response is 'Set1,Setup_2'.

```
MMEM:STOR:STAT 1, 'C:\Rohde&Schwarz\NWA\RecallSets\Setup_2.zvx';
MEM:DEL 'Setup_2.zvx'
```

Store the active setup Setup_2 to a file, renaming it Setup_2.zvx. Close the setup.

### MEMory:DEFine '<setup_name>'

Creates a new setup <setup_name> using default settings for the traces, channels and diagram areas. The created setup becomes the active setup.

**'<setup_name>'** String parameter to specify the name of the created setup.

*RST value –

**SCPI, Command Types** Device-specific, no query.

**Example:**

```
See MEM:CAT?.
```

### MEMory:DELete[:NAME] '<file_name>'

Closes the specified setup.

| | |
|---|---|
| **'<setup_name>'** | String parameter to specify the name of the setup to be closed. |
| *RST value | – |
| **SCPI, Command Types** | Confirmed, no query. |
| **Example:** | `See MEM:CAT?.` |

### MEMory:SELect '<setup_name>'

Selects a setup as the active setup.

| | |
|---|---|
| **'<setup_name>'** | String parameter to specify the setup. |
| *RST value | – |
| **SCPI, Command Types** | Device-specific, no query. |
| **Example:** | `*RST; MEM:DEF 'SETUP_2'` |
| | `Create a setup named 'Setup_2' and make it the active setup.` |
| | `MEM:SEL 'Set1'` |
| | `Activate the default setup Set1.` |
| | `MMEM:STOR:STAT 1, 'C:\Rohde&Schwarz\NWA\RecallSets\Set1.zvx'; MEM:DEL 'Set1.zvx'` |
| | Store the active setup Set1 to a file, renaming it Set1.zvx. Close the setup. |

## MMEMory...

The MMEMory system provides mass storage capabilities for the analyzer.

**Internal and external mass storage**

The mass storage of the analyzer may be internal or external. The internal mass storage device can be any section of the internal hard disk (mapped to drive C:\). The external mass storage device can be a floppy disk inserted into the drive at the front panel of the instrument, which is mapped to drive a:\ (see `MMEMory:MSIS`), 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 <file_name> and <directory_name> parameters are strings. Some commands use a fixed directory; for others the <file_name> can contain the complete path including the drive name and all subdirectories, e.g. 'C:\TEMP\TRASH\test.txt' for the file named test.txt in the TEMP\TRASH subdirectory of the internal hard disk drive C:\. If no complete path is specified, the file location is relative to the current directory (queried with `MMEMory:CDIRectory?`). The file name itself may contain the period as a separator for extensions.

File and directory names can be chosen according to Windows™ conventions; the restrictions placed on file names known from DOS systems do not apply. All letters and numbers are allowed, as well as 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.

## MMEMory:AKAL:FACTory:CONVersion '<directory_name>'

Converts the 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.

**'<directory_name>'** String parameter to specify the directory.

*RST value –

**SCPI, Command Types** Device-specific, command or query (returns the current directory).

**Example:**

```
MMEM:AKAL:FACTory:CONVersion 'C:\AKAL\Touchstone'
Convert and copy the calibration data of the standards
to the specified directory.
```

## MMEMory:CATalog? [<directory_name>]

Returns the contents of the current or of a specified directory.

Use MMEMory:CATalog:ALL? to query the contents of the current directory and all subdirectories.

**'<directory_name>'** 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?`

**Response** Directory information in the following format:

```
<used_size>,<free_disk_space>
{,<file_name>,,<file_size>}
```

The first two numeric parameters denote the total amount of storage currently used in the directory and the total amount of storage available, both expressed in bytes. All files are returned with their file name and their size in bytes.

*RST value –

**SCPI, Command Types** Confirmed, query only.

**Example:**

```
MMEM:CAT?
Response:  2056878, 809734144, zvb_quickstart.pdf, , 2056878,
```

## MMEMory:CATalog:ALL? [<directory_name>]

Returns the contents of the current or of a specified directory and all subdirectories.

Use MMEMory:CATalog? to query the contents of the current directory.

| | |
|---|---|
| **'<directory_name>'** | 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?` |
| **Response** | Directory information in the following format:<br><br>`Directory of <Dir_name>, <used_size>, <free_disk_space>`<br>`{, <Subdir>, <DIR>,}{,<file_name>,,<file_size>},`<br>`Directory of <Subdir_name_1>,`<br>`<used_size>,<free_disk_space> {, <Subdir>,`<br>`<DIR>,}{,<file_name>,,<file_size>},`<br>`Directory of <Subdir_name_2>,`<br>`<used_size>,<free_disk_space> ...`<br><br>The first line is omitted if the current directory contains no subdirectories. The following lines are analogous to the output of MMEMory:CATalog?. |
| *RST value | – |
| **SCPI, Command Types** | Confirmed, query only. |
| **Example:** | `MMEM:CAT:ALL?`<br><br>`Response:` Directory of C:\Rohde&Schwarz\Nwa\DOC\, 2056878, 809734144, zvb_quickstart.pdf, , 2056878, |

## MMEMory:CDIRectory '<directory_name>'

Changes the default directory for mass memory storage.

| | |
|---|---|
| **'<directory_name>'** | String parameter to specify the directory. |
| *RST value | – (A *RST does not change the current directory) |
| **SCPI, Command Types** | Confirmed, command or query (returns the current directory). |
| **Example:** | `MMEM:CDIR 'C:\Documents and Settings\NetworkService\Application Data'`<br><br>`Change to the specified directory.` |

## MMEMory:COPY '<file_source>','<file_destination>'

Copies an existing file to a new file.

| | |
|---|---|
| **'<file_source>', '<file_destination>'** | String parameters to specify the name of the file to be copied and the name of the new file. |

| | |
|---|---|
| *RST value | – |
| **SCPI, Command Types** | Confirmed, no query. |
| **Example:** | `MMEM:COPY 'C:\USER\DATA\SETUP.zvx','A:'`<br>`Copy file Setup.zvx in directory C:\USER\DATA to the external storage medium, mapped to drive A:\.` |

## MMEMory:DATA '<file_name>', <data>

Loads <data> into the file <file_name>.

| | |
|---|---|
| **'<file_name>'** | String parameter to specify the name of the file. |
| **<data>** | Data in 488.2 block data format. The delimiter EOI must be selected to achieve correct data transfer. |
| *RST value | – |
| **SCPI, Command Types** | Confirmed, with query. The query form is `MMEMory:DATA? <file_name>` with the response being the associated <data> in block format. |
| **Example:** | `MMEM:DATA? 'C:\TEMP\TEST01.HCP'`<br>`Query the block data contained in file TEST01.HCP.` |

## MMEMory:DELete '<file_name>'

Removes a file from the specified directory.

| | |
|---|---|
| **'<file_name>'** | String parameter to specify the name and directory of the file to be removed. |
| *RST value | – |
| **SCPI, Command Types** | Confirmed, no query. |
| **Example:** | `MMEM:DEL 'C:\TEMP\TEST01.HCP'`<br>`Remove TEST01.HCP from the current directory.` |

## MMEMory:DELete:CORRection <file_name>'

Deletes a system error correction data set stored in the cal pool (cal group file).

| | |
|---|---|
| **'<file_name>'** | 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:\Rohde&Schwarz\NWA\Calibration\Data.* |

| | |
|---|---|
| *RST value | – |
| **SCPI, Command Types** | Device-specific, no query. |
| **Example:** | `See MMEMory:LOAD:CORRection` |

## MMEMory:LOAD:CKIT '<file_name>'

Loads cal kit data from a specified NWA cal kit file.

| | |
|---|---|
| **'<file_name>'** | String parameter to specify the name and directory of the cal kit file to be loaded. If no path is specified the analyzer searches the current directory, to be queried with `MMEMory:CDIRectory?`.<br>*Note: The loaded file must be a NWA-specific cal kit file with the extension *.calkit. ZVR cal kit files can be imported using the `[SENSe<Ch>:]CORRection:CKIT:INSTall` command. Agilent cal kit files can be imported manually and converted into *.calkit files.* |
| *RST value | – |
| **SCPI, Command Types** | Device-specific, no query. |
| **Example:** | `MMEM:LOAD:CKIT 'C:\Rohde&Schwarz\NWA\Calibration\Kits\New_kit.calkit'`<br>Load the previously created cal kit file New_kit.calkit from the default cal kit directory.<br>`... :MMEM:STOR:CKIT 'New_kit', 'C:\Rohde&Schwarz\NWA\Calibration\Kits\New_kit.calkit'`<br>Store the data for the user-defined cal kit Newkit and overwrite the cal kit file New_kit.calkit. |

## MMEMory:LOAD:CKIT:SDATa '<conn_name>', '<ckit_name>', MMTHrough | MFTHrough | FFTHrough | MMLine | MFLine | FFLine | MMATten | MFATten | FFATten | MMSNetwork | MFSNetwork | FFSNetwork | MOPen | FOPen | MSHort | FSHort | MOSHort | FOSHort | MREFlect | FREFlect | MMTCh | FMTCh | MSMatch | FSMatch, '<stdlabel_name>', '<file_name>' [,<port1_no>][,<port2_no>]

Loads cal kit data for a specific connector type, cal kit, and calibration standard from a specified Touchstone file, assigning a label for the cal data. A restriction on the port assignment may be defined in addition.

| | |
|---|---|
| **'<conn_name>'** | String parameters containing the name of the connector type. |
| **'<ckit_name>'** | String parameters containing the name of a calibration kit available on the analyzer. |
| **Parameters** | Standard type; for a description refer to the table of standard types. |

| | |
|---|---|
| **'<stdlabel_name>'** | String parameter defining a label for the cal kit data. An empty string means that no label is defined. |
| **'<file_name>'** | 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?`. |
| **<port1_no>, <port2_no>** | Port assignment: One port number for one-port standards, two port number for two-port standards. If the port numbers are omitted, the cal kit data is valid for all ports. |
| *RST value | – |
| **SCPI, Command Types** | Device-specific, no query. |
| **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 data standard data is only valid for port no. 1. |

**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 Dummy Kit*.
- 'ZV-Z21 typical' denotes the cal kit *ZV-Z21 typical*.

## MMEMory:LOAD:CMAP '<file_name>'

Loads a color scheme from a specified NWA color scheme file.

| | |
|---|---|
| **'<file_name>'** | String parameter to specify the name and directory of the cal kit file to be loaded. The default extension (manual control) for color scheme files is *.ColorScheme, although other extensions are allowed. |
| *RST value | – |
| **SCPI, Command Types** | Device-specific, no query. |
| **Example:** | `MMEM:LOAD:CMAP 'C:\Rohde&Schwarz\NWA\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:\Rohde&Schwarz\NWA\ColorSchemes\Test.ColorScheme'
```

```
Store the data for the user-defined cal kit Newkit and
overwrite the cal kit file New_kit.calkit.
```

## MMEMory:LOAD:CORRection <Ch>,'<file_name>'

Applies a system error correction data set stored in the cal pool (cal group file) to channel no. <Ch>.

**<Ch>** Channel number of an existing channel

**<file_name>** 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:\Rohde&Schwarz\NWA\Calibration\Data.*

*RST value –

**SCPI, Command Types** Device-specific, no query.

**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 to channel 2.
```

```
MMEM:LOAD:CORR:RES 2,'Calgroup1.cal'
```

```
Undo the previous action: Resolve the link, causing channel
2 to use its previous correction data.
```

```
MMEM:DEL:CORR 'Calgroup1.cal'
```

```
Delete the created cal group file.
```

## MMEMory:LOAD:CORRection:RESolve <Ch>[,'<file_name>']

Resolves the link between channel <Ch> and the correction data set (cal group file) so that the channel uses its previous correction data.

**<Ch>** Channel number

**'<file_name>'** Optional string parameter to specify the name of the cal group file. Cal group files must have the extension *.cal. The directory path must not be specified. 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.

*RST value –

**SCPI, Command Types** Device-specific, no query.

**Example:**

```
See MMEMory:LOAD:CORRection
```

## MMEMory:LOAD:LIMit '<trc_name>','<file_name>'[,'<param_name>', <x_offset>, <y_offset>, <type>]

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<Chn>:LIMit...` commands.

*Limit lines can be loaded from Touchstone files (*.s<n>p, where <n> denotes the number of ports). The optional parameters '<param_name>', <x_offset>, <y_offset>, <type> are only relevant for Touchstone files. For *.limit files, no optional parameters can be set.*

**'<trc_name>'** Name of an existing trace in the active setup (string parameter). The imported limit line is assigned to this trace, irrespective of the trace information in the limit line file.

*RST value –

**'<file_name>'** 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.

*RST value –

**'<param_name>'** 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 value 'S11' (if all optional parameters are omitted)

**<x_offset>** 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 [def. unit] Depending on the sweep range of the analyzer. [The default units for frequency, power and time sweeps are Hz, dB and s, respectively.]

*RST value 0

**<y_offset>** 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 [def. unit] Depending on the measured quantity. [dB]

*RST value 0

**<type>** Limit line type :
LMAX – upper limit line
LMIN – lower limit line
OFF – limit line off

*RST value LMAX (if all optional parameters are omitted)

**SCPI, Command Types** Device-specific, no query.

**Example:**

```
Assume that the current setup contains two traces named
```

Trc1 and Trc2, respectively, and that limit lines have been defined for Trc1.

```
MMEM:STOR:LIM 'TRC1',
'C:\Rohde&Schwarz\NWA\LIMitLines\Lim_Trc1.limit'
```

Store the limit line definition of Trc1 to a limit line file.

```
MMEM:LOAD:LIM 'TRC2',
'C:\Rohde&Schwarz\NWA\LIMitLines\Lim_Trc1.limit'
```

Load the previously created limit line file and assign the limit lines to Trc2.

```
MMEM:STOR:TRAC 'TRC1',
'C:\Rohde&Schwarz\NWA\LIMitLines\Trc1.s1p'
```

Store the current trace data of Trc1 to a limit line file in Touchstone format.

```
MMEM:LOAD:LIM 'TRC1',
'C:\Rohde&Schwarz\NWA\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.

## MMEMory:LOAD:SEGMent <Ch>,'<file_name>'

Loads a sweep segment definition from a specified file and assigns it to a specified channel. Sweep segments are defined using the `[SENSe<Ch>:]SEGMent<Seg>...` commands.

**<Ch>** Channel number.

**'<file_name>'** 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?`.

*RST value –

**SCPI, Command Types** Device-specific, no query.

**Example:**

Assume that the current setup contains two channels numbered 1 and 2, respectively, and that sweep segments have been defined for channel no. 1.

```
:MMEM:STOR:SEGM 1,
'C:\Rohde&Schwarz\NWA\SweepSegments\Seg_Ch1.seglist'
```

Store the sweep segment definition of channel 1 to a sweep segment file.

```
MMEM:LOAD:SEGM 2,
'C:\Rohde&Schwarz\NWA\SweepSegments\Seg_Ch1.seglist'
```

Load the previously created sweep segment file and use the sweep segments for channel 2.

## MMEMory:LOAD:STATe <numeric_value>,'<file_name>'

Loads configuration data from a specified setup file and sets the analyzer to the corresponding instrument state.

**<numeric_value>** 1 (the <numeric_value> is used for compatibility with the SCPI standard but is ignored).

**<file_name>** String parameter to specify the name and directory of the setup file to be loaded. The default extension (manual control) for setup files is *.zvx, although other extensions are allowed. If no path is specified the analyzer searches the current directory, to be queried with `MMEMory:CDIRectory?`.

*RST value –

**SCPI, Command Types** Confirmed, no query.

**Example:**

```
MMEM:STOR:STAT
1,'C:\Rohde&Schwarz\NWA\RecallSets\Setup_0413.zvx'
```

Store the current setup configuration in the file Setup_0413.zvx in the default directory for setup files.

```
MMEM:LOAD:STAT
1,'C:\Rohde&Schwarz\NWA\RecallSets\Setup_0413.zvx'
```

Load the settings stored in Setup_0413.zvx.

## MMEMory:LOAD:TRACe '<trc_name>','<file_name>'[,'<parameter_name>']

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.

**'<trc_name>'** Name of an existing data trace in the active setup (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.

**'<file_name>'** String parameter to specify the name and directory of the trace file to be loaded. Several file formats for trace files are supported. 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 *.dat for Touchstone, ASCII, and Matlab files are mandatory.

**'<parameter_name>'** Optional string parameter for imported Touchstone files for more than one port (*.s2p, *.s3p, *.s4p), denotes the imported S-parameter ('S11', 'S12', ...). If nothing is specified, the first trace in the specified file is imported.

*RST value –

**SCPI, Command Types** Confirmed, no query.

**Example:**

```
Assume that the current setup contains a trace named
Trc1.
```

```
MMEM:STOR:TRAC 'TRC1',
'C:\Rohde&Schwarz\NWA\Traces\Trc1.s1p'
```

```
Store the current trace data of Trc1 to a trace file.
```

```
MMEM:LOAD:TRAC 'TRC1',
'C:\Rohde&Schwarz\NWA\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:\Rohde&Schwarz\NWA\Traces\Trc1.s2p'
```

```
Store the four S-parameter traces to a two-port
Touchstone file.
```

```
MMEM:LOAD:TRAC 'TRC1',
'C:\Rohde&Schwarz\NWA\Traces\Trc1.s2p'
```

```
Load the previously created Touchstone file and
overwrite the previously generated memory trace
assigned to Trc1 with the S11 trace.
```

## MMEMory:LOAD:VNETworks<Ch>:BALanced:DEEMbedding<Log_pt> '<file_name>', PMAin | PSECondary

Loads data from a specified Touchstone file defining the characteristics of the current deembedded balanced port circuit model. A balanced port circuit model involving file import must be selected before using the command.

**<Ch>** Channel number

**<Log_pt>** Logical port number

**'<file_name>'** String parameter to specify the name and directory of the loaded Touchstone file. The balanced port circuit models `STSL | STSC | SLST | SCST` require two 2-port (*.s2p) files, to be assigned to the different ports `PMAin` and `PSECondary`; the `FIMPort` model requires a single 4-port (*.s4p) file but no additional port assignment.
If no path is specified the analyzer searches the current directory, to be queried with `MMEMory:CDIRectory?`.

**Parameters** Port assignment for two 2-port (*.s2p) files:
PMAin – Port 1
PSECondary – Port 2
The ports **must be specified** for the import of 2-port (*.s2p) files; they **must be omitted** for 4-port (*.s4p) files.

*RST value –

**SCPI, Command Types** Device-specific, no query.

**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:\Rohde&Schwarz\NWA\VNETWORKS\Test.s2p', PMA
```

Load a Touchstone file and assign it to logical port no. 2.

## MMEMory:LOAD:VNETworks<Ch>:BALanced:EMBedding<Log_pt> '<file_name>', PMAin | PSECondary

Loads data from a specified Touchstone file defining the characteristics of the current embedded balanced port circuit model. A balanced port circuit model involving file import must be selected before using the command.

**<Ch>** Channel number

**<Log_pt>** Logical port number

**'<file_name>'** String parameter to specify the name and directory of the loaded Touchstone file. The balanced port circuit models `STSL | STSC | SLST | SCST` require two 2-port (*.s2p) files, to be assigned to the different ports `PMAin` and `PSECondary`; the `FIMPort` model requires a single 4-port (*.s4p) file but no additional port assignment.
If no path is specified the analyzer searches the current directory, to be queried with `MMEMory:CDIRectory?`.

**Parameters** Port assignment for two 2-port (*.s2p) files:
PMAin – Port 1
PSECondary – Port 2
The ports **must be specified** for the import of 2-port (*.s2p) files; they **must be omitted** for 4-port (*.s4p) files.

*RST value –

**SCPI, Command Types** Device-specific, no query.

**Example:**

```
*RST; SOUR:LPOR1 1,2; LPOR2 3,4
```

Define a balanced port configuration.

```
CALC:TRAN:VNET:BAL:EMB:TND STSL
```

Select the *Serial .s2p data, shunt L* circuit model.

```
MMEM:LOAD:VNET:BAL:EMB2
'C:\Rohde&Schwarz\NWA\VNETWORKS\Test.s2p', PMA
```

Load a Touchstone file and assign it to logical port no. 2.

## MMEMory:LOAD:VNETworks<Ch>:SENDed:DEEMbedding<Ph_pt> '<file_name>'

Loads data from a specified two-port (*.sp2) Touchstone file defining the characteristics of the current deembedded single ended port circuit model. The single ended port circuit model involving file import (`FIMPort`) must be selected before using the command.

**<Ch>** Channel number.

**<Ph_pt>** Physical port number

**'<file_name>'** String parameter to specify the name and directory of the loaded Touchstone file. If no path is specified the analyzer searches the current directory, to be queried with `MMEMory:CDIRectory?`.

*RST value –

**SCPI, Command Types** Device-specific, no query.

**Example:**

```
CALC:TRAN:VNET:SEND:DEEM:TND FIMP
```

Select the *Serial .s2p data, shunt L* circuit model.

```
MMEM:LOAD:VNET:SEND:DEEM2
'C:\Rohde&Schwarz\NWA\VNETWORKS\Test.s2p'
```

Load a Touchstone file and assign it to the physical port no. 2.

## MMEMory:LOAD:VNETworks<Ch>:SENDed:EMBedding<Ph_pt> '<file_name>'

Loads data from a specified two-port (*.sp2) Touchstone file defining the characteristics of the current embedded single ended port circuit model. The single ended port circuit model involving file import (`FIMPort`) must be selected before using the command.

**<Ch>** Channel number.

**<Ph_pt>**

**'<file_name>'** String parameter to specify the name and directory of the loaded Touchstone file. If no path is specified the analyzer searches the current directory, to be queried with `MMEMory:CDIRectory?`.

*RST value –

**SCPI, Command Types** Device-specific, no query.

**Example:**

```
CALC:TRAN:VNET:SEND:EMB:TND FIMP
```

Select the *Serial .s2p data, shunt L* circuit model.

```
MMEM:LOAD:VNET:SEND:EMB2
'C:\Rohde&Schwarz\NWA\VNETWORKS\Test.s2p'
```

Load a Touchstone file and assign it to the physical port

```
no. 2.
```

## MMEMory:MDIRectory '<directory_name>'

Creates a new subdirectory for mass memory storage in an existing directory.

**'<directory_name>'** String parameter to specify the new directory. Either the full path or a subdirectory for the current directory (see `MMEMory:CDIRectory`).

*RST value –

**SCPI, Command Types** Device-specific, no query.

**Example:**

```
MMEM:MDIR 'C:\Documents and Settings\New_Directory'
```

```
Create the specified directory. The parent directory
'C:\Documents and Settings' must have been created
before.
```

```
MMEM:MDIR 'C:\Documents and
Settings\New_Directory\New_Subdirectory'
```

```
Create an additional subdirectory.
```

```
MMEM:CDIR 'C:'; MDIR 'New_Rootdirectory'
```

```
Create an additional directory C:\New Rootdirectory.
```

## MMEMory:MOVE '<file_source>','<file_destination>'

Moves a file to the indicated directory and stores it under the file name specified, if any. If <file_destination> contains no path indication, the command renames the file without moving it.

**'<file_source>', '<file_destination>'** 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.

*RST value –

**SCPI, Command Types** Confirmed, no query.

**Example:**

```
MMEM:MOVE 'C:\USER\DATA\SETUP.zvx','A:'
```

```
Move file Setup.zvx in directory C:\USER\DATA to the
external storage medium, mapped to drive A:\.
```

## MMEMory:NAME '<file_name>'

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'`).

**'<file_name>'** 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 value 'Hardcopy'

**SCPI, Command Types** Device-specific, with query.

**Example:**

```
MMEM:NAME 'C:\Screenshots\PLOT1.BMP'
```

```
Define a printer file name, to be stored in the existing
directory C:\Screenshots (without creating the file).
```

```
HCOP:DEST 'MMEM'; HCOP
```

```
Select 'Print to file' and create the printer file
specified before.
```

## MMEMory:RDIRectory '<directory_name>'

Removes an existing directory from the mass memory storage system.

**'<directory_name>'** String parameter to specify the directory.

*RST value –

**SCPI, Command Types** Device-specific, no query.

**Example:**

```
MMEM:RDIR 'C:\Documents and
Settings\NetworkService\Application Data'
```

```
Removes the specified directory.
```

## MMEMory:STORe:CKIT '<kit_name>', '<file_name>'

Stores the data of a calibration kit to a specified file.

**'<kit_name>'** Name of a user-defined calibration kit available on the analyzer.

NOTE *It is not possible to modify or store predefined or ideal kits.*

**'<file_name>'** 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 uses the current directory, to be queried with `MMEMory:CDIRectory?`.

*RST value –

**SCPI, Command Types** Device-specific, no query.

**Example:**

```
MMEM:LOAD:CKIT
'C:\Rohde&Schwarz\NWA\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:\Rohde&Schwarz\NWA\Calibration\Kits\New_kit.calkit'

Store the data for the user-defined cal kit Newkit and
overwrite the cal kit file New_kit.calkit.
```

## MMEMory:STORe:CMAP '<file_name>'

Stores a color scheme to a specified NWA color scheme file.

**'<file_name>'** 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.

*RST value –

**SCPI, Command Types** Device-specific, no query.

**Example:**

```
See MMEMory:LOAD:CMAP
```

## MMEMory:STORe:CORRection <Ch>,'<file_name>'

Copies the correction data of channel <Ch> to the cal pool, generating a new correction data file (cal group). The file has the extension *.calkit and is stored in the *C:\Rohde&Schwarz\NWA\Calibration\Data* directory.

**<Ch>** Channel number

**'<file_name>'** String parameter to specify the name of the created cal group file. The directory path must not be specified; the analyzer always uses the default cal pool directory *C:\Rohde&Schwarz\NWA\Calibration\Data.*

*RST value –

**SCPI, Command Types** Device-specific, no query.

**Example:**

```
See MMEMory:LOAD:CORRection
```

## MMEMory:STORe:LIMit '<trc_name>', '<file_name>'

Saves the limit lines associated to a specified trace to a limit line file. Limit lines are created using the `CALCulte<Chn>:LIMit...` commands.

**'<trc_name>'** Name of an existing trace in the active setup (string parameter) for which a limit

line definition exists.

**'<file_name>'** 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 uses the current directory, to be queried with `MMEMory:CDIRectory?`.

*RST value –

**SCPI, Command Types** Device-specific, no query.

**Example:**

```
See MMEMory:LOAD:LIMit.
```

## MMEMory:STORe:MARKer '<file_name>'

Saves the values of all markers to a ASCII file.

**'<file_name>'** 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 uses the current directory, to be queried with `MMEMory:CDIRectory?`. An example for the file contents is given below.

*RST value –

**SCPI, Command Types** Device-specific, no query.

**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:
Trc1 S21
Mkr 1 1.000000 GHz  -4.900 dB
Mkr 2 2.000000 GHz  -6.087 dB
```

## MMEMory:STORe:SEGMent <Ch>, '<file_name>'

Saves the sweep segment definition of a specified channel to a sweep segment file. Sweep segments are defined using the `[SENSe<Ch>:]SEGMent<Seg>...` commands.

**<Ch>** Channel number.

| | |
|---|---|
| **'<file_name>'** | 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 uses the current directory, to be queried with `MMEMory:CDIRectory?`. |
| *RST value | – |
| **SCPI, Command Types** | Device-specific, no query. |
| **Example:** | `See MMEMory:LOAD:SEGMent.` |

## MMEMory:STORe:STATe <numeric_value>,'<file_name>'

Stores the configuration data of the current setup to a specified setup file.

*`MMEMory:STORe:STATe` renames the current setup, appending a .zvx extension. See example for `MMEMory:LOAD:STATe`.*

| | |
|---|---|
| **<numeric_value>** | 1 (the <numeric_value> is used for compatibility with the SCPI standard but is ignored). |
| **'<file_name>'** | String parameter to specify the name and directory of the created setup file. The default extension (manual control) for setup files is *.zvx, although other extensions are allowed. If no path is specified the analyzer uses the current directory, to be queried with `MMEMory:CDIRectory?`. |
| *RST value | – |
| **SCPI, Command Types** | Confirmed, no query. |
| **Example:** | `See MMEMory:LOAD:STATe.` |

## MMEMory:STORe:TRACe '<trc_name>','<file_name>'[,UNFormatted | FORMatted, COMPlex | LINPhase | LOGPhase]

Stores the trace data of a specified trace to a trace file. Traces are created using the `CALCulte<Ch>:PARameter:SDEFine...` command.

*.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. If the necessary trace are available, '<trc_name>' can be the name of any of the traces.

| | |
|---|---|
| **'<trc_name>'** | Name of an existing data trace in the active setup (string parameter). |
| **'<file_name>'** | 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.<br>If no path is specified the analyzer uses the<br>`C:\Rohde&Schwarz\NWA\Traces` directory. |
| **Optional parameters** | UNFormatted – Unformatted data export specified by the second optional parameter<br>FORMatted – Formatted data export (for *.csv and *.dat files only)<br>If the first optional parameter is omitted the command stores unformatted data.<br>COMPlex – Complex values (real and imaginary part)<br>LINPhase – Linear magnitude and phase<br>LOGPhase – dB-magnitude and phase<br>If the second optional parameter is omitted the command stores complex data. |
| *RST value | – (the default export format is UNFormatted) |
| **SCPI, Command Types** | Confirmed, no query. |
| **Example:** | `See MMEMory:LOAD:TRACe.` |

# OUTPut<Pt>...

This subsystem controls the characteristics of the analyzer's output ports.

## OUTPut<port_no>:ATTenuation

Sets an attenuation factor for the generated wave transmitted at test port no. <Pt>. The incident wave is attenuated via `INPut<Pt>:ATTenuation`.

*`OUTPut<Pt>:ATTenuation` is not channel-specific; the value is valid for all channels. Use `SOURce<Ch>:POWer<Pt>:ATTenuation` to set or query a channel-specific attenuation value.*

*Use SOURce<Ch>:POWer<Pt>:ATTenuation:AUTO ON for automatic generator attenuator setting. Automatic attenuator setting overwrites the manual attenuation factor. Conversely, a manual entry resets the mode to manual (OFF).*

| | |
|---|---|
| **<port_no>** | Test port number of the analyzer, 1 to 2, 3 or 4, depending on the analyzer model. If unspecified the numeric suffix is set to 1. |
| **<numeric_value>** | Attenuation factor for the generated wave. |
| Range [def. unit] | Depending on the analyzer/attenuator model, e.g. 0 dB to 80 dB for ZVA8 with ZVA8-B21/B22/B23/B24 [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 value | 0 dB |
| **SCPI, Command Types** | Confirmed, no query (query can not return channel-specific settings, would be misleading). |
| **Example:** | `SOUR:POW -6`<br>Set the internal source power for channel 1 to -6 dBm.<br>`OUTP2:ATT 10`<br>Set the step attenuator for the wave transmitted at port 2 and for all channels to 10 dB. The internal source power and the waves at the other test ports |

are not affected.

```
SOUR1:POW2:ATT?
```

Query the generator step attenuator setting at port 2 and for channel no. 1. The response is *10*.

## OUTPut<Chn>:DPORt PORT1 | PORT2 | PORT3 | PORT4

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:*** *This command is equivalent to* `[SENSe<Chn>:]SWEep:SRCPort`.

**<Chn>** Channel number used to identify the active trace. If unspecified the numeric suffix is set to 1.

**PORT1 ...** Test port number of the analyzer, 1 to 2, 3 or 4, depending on the analyzer model.

*RST value PORT1

**SCPI, Command Types** Device-specific, with query.

**Example:**

```
CALC4:PAR:SDEF 'Ch4Tr1', 'A1'
```

Create channel 4 and a trace named *Ch4Tr1* to measure the wave quantity a1. The trace automatically becomes the active trace.

```
OUTP4:DPOR PORT2
```

Select drive port 2 for the active trace.

## OUTPut<Ch>[:STATe] <Boolean>

Turns the internal source power at all ports and the power of all external generators on or off.

*After the RF power is switched on, the start of the measurement is delayed by approx. 1 s while the amplifier stages of the analyzer are allowed to settle. To circumvent this delay if you need to turn off the base channel power for a limited period of time, reduce the base channel power setting or turn off the drive ports in the Port Configuration dialog instead of using RF Off.*

**<Ch>** Channel number. This suffix is ignored; the setting is valid for all channels.

**<Boolean>** ON | OFF - Switches the power on or off.

*RST value ON

**SCPI, Command Types** Device-specific, command or query

**Example:**

```
OUTP OFF
```

Turn off the RF source power.

## OUTPut<Ch>:UPORt[:VALue] <numeric_value>

Sets or queries a channel-dependent four-bit binary value to control four independent output signals at the USER CONTROL connector (lines 8, 9, 10, 11). The output signals are 3.3 V TTL signals which can be used to differentiate between up to 16 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 of the USER CONTROL connector.
- The value defined with `OUTPut<Ch>:UPORt[:VALue]` is assigned to channel no. <Ch>.
- The signals at the USER CONTROL 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 CONTROL 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.

You can use the active channel number as a parameter for `OUTPut<Ch>:UPORt[:VALue]` and monitor the activity of up to 16 different channels at the USER CONTROL connector; see example below. You can also use the USER CONTROL output signals as channel-dependent trigger signals for external devices. Use `CONTrol:AUXiliary:C[:DATA]` to transfer the four bit value in decimal representation.

| | |
|---|---|
| **<Ch>** | Channel number. |
| **<numeric_value>**<br>Range [def. unit] | Binary value.<br>#B0000 to #B1111 (for setting command), 0 to 15 (query) [–]<br>The transferred values correspond to the following states of the USER CONTROL connector:<br>#B0000 – no signal at any of the four pins 8, 9, 10, 11<br>#B0001 – output signal at pin 8<br>#B0010 – output signal at pin 9<br>#B0011 – output signal at pin 8 and 9<br>...<br>#B1111 – output signal at pin 8, 9, 10 and 11 |
| *RST value | #B0000 (0) |
| **SCPI, Command** | Confirmed, command or query. The query returns a decimal value. |

**Types**

**Example:**

```
*RST; OUTP1:UPOR #B0001
```

Assign the channel bit value #B0001 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 #B0020
```

Create channel no. 2, causing it to become the active channel, and assign the channel bit value #B0002. 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.

# PROGram...

This subsystem controls external application programs that can be run on the analyzer.

## PROGram[:SELected]:EXECute '<file_name>[ <command>]'

Starts an application program or open a file using an application available on the analyzer.

**'<file_name>'** 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 file name are not allowed because everything after a blank is interpreted as a <command>.

**<command>** Command name in the selected application <file_name> (optional).

*RST value –

**SCPI, Command**

**Types** Confirmed, no query.

**Example:**

```
PROG:SEL:NAME  PROG
```

Selects general program execution on the analyzer.

```
MMEM:CDIR 'C:\Program Files\Rohde&Schwarz\Network Analyzer\Bin'
```

Go to the program directory.

```
PROG:EXEC 'iecwin32.exe'
```

Start the application iecwin32.

```
PROG:EXEC:WAIT
```

Lock command execution and manual control of the analyzer until iecwin32 is paused or stopped.

### PROGram[:SELected]:NAME PROG

Selects the application to be run on the analyzer. At present, only the general parameter PROG is available: `PROGram[:SELected]:EXECute` can start any program.

Use this command in order to avoid problems should the default value change in future firmware versions.

| | |
|---|---|
| **PROG** | Any program running under Windows XP or any file that can be opened with an application program available on the analyzer. |
| *RST value | PROG |
| **SCPI, Command Types** | Confirmed, command or query. |
| **Example:** | `See PROGram[:SELected]:EXECute.` |

### PROGram[:SELected]:WAIT

Locks command execution and manual control of the analyzer 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.

| | |
|---|---|
| **Parameters** | None. |
| *RST value | – |
| **SCPI, Command Types** | Confirmed, command or query. The query also locks command execution; it returns 1 when the executed program is stopped or paused. |
| **Example:** | `See PROGram[:SELected]:EXECute.` |

## [SENSe...]

### [SENSe<Ch>:]AVERage...

This subsystem sets 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.

### [SENSe<Ch>:]AVERage:CLEar

Starts a new average cycle, clearing all previous results and thus eliminating their effect on the new cycle.

| | |
|---|---|
| **<Ch>** | Channel number. If unspecified the numeric suffix is set to 1. |
| *RST value | – |
| **SCPI, Command** | Confirmed, no query |

**Types**

**Example:**

```
AVER:COUN 15; AVER ON
```

Set the average factor for channel 1 to 15 and enable the sweep average.

```
AVER:COUN 5; CLE
```

Reduce the average factor and restart the average.

## [SENSe<Ch>:]AVERage:COUNt <numeric_value>

Defines the number of consecutive sweeps to be combined for the sweep average *(Average Factor)*.

**<Ch>** Channel number.

**<numeric_value>**
Range [def. unit]
*RST value

Average factor
1 to 1000 [1]
10

**SCPI, Command Types** Confirmed, command or query

**Example:**

```
AVER:COUN 15
```

Set the average factor for channel 1 to 15.

```
AVER ON
```

Enable the sweep average.

## [SENSe<Ch>:]AVERage[:STATe] <Boolean>

Enable or disable the sweep average.

**<Ch>** Channel number.

**<Boolean>** ON | OFF - Enables or disables the automatic calculation of the sweep average over the specified number of sweeps ([SENSe<Ch>:]AVERage:COUNt).

*RST value ON

**SCPI, Command Types** Confirmed, command or query

**Example:**

```
AVER:COUN 15
```

Set the average factor for channel 1 to 15.

```
AVER ON
```

Enable the sweep average over the defined number of sweeps.

## [SENSe<Ch>:]BANDwidth...

This subsystem sets the bandwidth of the IF measurement filter (resolution bandwidth).The forms `BANDwidth` and `BWIDth` are equivalent.

## [SENSe<Ch>:]BANDwidth|BWIDth[:RESolution] <bandwidth>

Defines the resolution bandwidth of the analyzer *(Meas. Bandwidth)*.

| | |
|---|---|
| **<Ch>** | Channel number. If unspecified the numeric suffix is set to 1. |
| **<bandwidth>** | Resolution bandwidth |
| Range [def. unit] | 1 Hz to 1 MHz (5 (10) MHz with option R&S ZVA-K17, 10 (30) MHz with option R&S ZVA-K7; see *Pulsed Measurements*) [Hz]. UP and DOWN increment/decrement the bandwidth in 1-2-5 steps for each decade. The analyzer rounds up any entered value between these steps and rounds down values exceeding the maximum bandwidth. |
| *RST value | 10 kHz |
| **SCPI, Command Types** | Confirmed, command or query |
| **Example:** | `BAND 1.1` |
| | Set a resolution bandwidth of approx. 1.1 Hz for channel 1. |
| | `BAND?` |
| | The analyzer returns the rounded bandwidth of 2 Hz. |

## [SENSe<Ch>:]BANDwidth|BWIDth[:RESolution]:SELect NORMal | HIGH

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]:CONTrol OFF).

| | |
|---|---|
| **<Ch>** | Channel number. |
| **NORMal** | Use IF filter with normal selectivity and short settling time. |
| **HIGH** | Use IF filter with higher selectivity but larger settling time. |
| *RST value | NORMal |

**SCPI, Command Types** Device-specific, command or query

**Example:** `See [SENSe<Ch>:]SEGMent<Seg>:BWIDth[:RESolution]:CONTrol`

## [SENSe<Ch>:]CORRection...

This subsystem controls system error correction and recording of correction data.

## [SENSe<Ch>:]CORRection:CDATa 'DIRECTIVITY' | 'SRCMATCH' | 'REFLTRACK' | 'ISOLATION' | 'LOADMATCH' | 'TRANSTRACK' | 'G11' | ... 'G22' | 'H11' | ... | 'H22', <port1_no>, <port2_no>

Writes or reads system error correction data for a specific channel <Ch>, calibration method ([SENSe<Ch>:]CORRection:COLLect:METHod:DEFine), and port combination <Port1_no>, <Port2_no>. 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:SELected:DEFault.*

For an overview of calibration methods and error terms refer to section Calibration Types.

**<Ch>** Channel number of the calibrated channel. If unspecified the numeric suffix is set to 1.

**Error term parameters** 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.

Range [def. unit] The error terms consist of dimensionless complex numbers. The parameters must be transferred in full length and have the following meaning:
'DIRECTIVITY' – Directivity at port <port1_no>
'SRCMATCH' – Source match at port <port1_no>
'REFLTRACK' – Reflection tracking at port <port1_no>
'ISOLATION' – Isolation between port <port1_no> and <port2_no>
'LOADMATCH' – Load match at <port2_no>
'TRANSTRACK' – Transmission tracking between port <port1_no> and <port2_no>
'G11' ... 'G22' – G matrix elements, referenced to <port1_no> (irrespective of the values of <port1_no> and <port2_no>)
'H11' ... 'H22' – H matrix elements, referenced to <port2_no> (irrespective of the values of <port1_no> and <port2_no>)

**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 $H_{21} = 1$.

*RST value –
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`.

**<port1_no>** Source port number.

**<port2_no>** 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`

**SCPI, Command Types** Device-specific, command or query

NOTE *After a system error correction in one of the sweep types Power, Time, or CW Mode, the query returns two sets of correction data, corresponding to the fixed CW frequency and the CW frequency plus 1 Hz (minus 1 Hz if the CW frequency is equal to the maximum frequency of the analyzer).*

**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 | 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' (at present the 'ISOLATION' terms are not included) |
| TOM, 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:CKIT...

This subsystem controls system error correction and recording of correction data.

## [SENSe<Ch>:]CORRection:CKIT:<std_type> '<Conn_Name>','<Ckit_Name>','<Std_No>',<Min_Freq>,<Max_Freq>,<El_Length>,<Loss>,<Z0>,<C0> , <C1>, <C2>, <C3>, <L0>, <L1>, <L2>, <L3>[, OPEN | SHORt | MATCh, <Resistance>, <Port_1>, <Port_2>]

Defines the parameters of a non-ideal 1 port or 2-port calibration standard <std_type>. 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 may be used; see table below.

**<Ch>** Channel number. This suffix is ignored because calibration kits are channel-independent.

**<std_type>** Standard type. For reflection standards, the first character denotes the gender, e.g.:
`FOPEN, MOPEN`: Open (f) or Open (m) standard.
The following reflection standards are supported: `MOPen, FOPen, MSHort, FSHort, OSHort, MOSHort, FOSHort, MMTCh, FFTCh, MREFLect, FREFLect`
For transmission standards, the first two characters denotes 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 the table of standard types below.

**Parameter list** String parameters to specify the configured standard ('<Conn_Name>','<Ckit_Name>','<Std_No>') and numeric parameters defining its properties. See detailed description below.

NOTE *The following transmission standards have only 8 parameters (until <Z0>):*

*MMTHrough, MFTHrough, FFTHrough, MMLine, MFLine, FFLine*
*The following reflection and transmission standards have only 5 parameters (until <Max_Freq>):*
*MSMatch, FSMatch, MMATten, MFATten, FFATten*

*RST value –

**SCPI, Command Types** Device-specific, command or query (see examples below)

**Example:**

```
CORR:CKIT:FOP  'N 50 Ohm','Test Kit',...
```

Define the properties of the open (f) standard for a N 50 Ω connector type in cal kit 'Test Kit'. See also example for `[SENSe<Ch>:]CORRection:CKIT:<conn_type>:<std_type>`.

```
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','Test Kit'
```

Query the properties of the open (f) standard for a N 50 Ω connector type in cal kit 'Test Kit'.

## [SENSe<Ch>:]CORRection:CKIT:<conn_type>:<std_type> '<Ckit_Name>','<Std_No>',<Min_Freq>,<Max_Freq>,<El_Length>,<Loss>,<C0> | <L0>,<C1> | <L1>,<C2> | <L2>,<C3> | <L3>, OPEN | SHORt

Defines the parameters of a calibration standard <std_type> for a specified connector type <conn_type>. 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 may be used; see table below.

**<Ch>** Channel number. This suffix is ignored because calibration kits are channel-independent.

**<conn_type>** 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<Ch>:]CORRection:CKIT:<std_type>`.*

**<std_type>** 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 denotes the genders on both ends, e.g.:
`FFTHrough, MFTHrough, MMTHrough`: Through (ff), Through (mf) or Through (mm) standard.
For a complete list of standard types refer to the table of standard types below.

**Parameter list** String parameters to specify the configured standard ('<Ckit_Name>','<Std_No>') and numeric parameters defining its properties. See detailed description below.

*RST value –

**SCPI, Command Types** Device-specific, command or query

**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)$^3$.

The parameters in the `[SENSe<Ch>:]CORRection:CKIT:<std_type>` and `[SENSe<Ch>:]CORRection:CKIT:<conn_type>:<std_type>` commands have the following meaning:

| Parameter | Meaning | Comment/Unit |
|---|---|---|
| '<Conn_Name>' | Name of the connector type | String parameter |
| '<Ckit_Name>' | Name of the calibration kit | String parameter |
| '<Std_No>' | Label (e.g. the serial number) of the standard | String parameter |
| <Min_Freq> | Min. frequency for which the circuit model is valid | Default unit is Hz |
| <Max_Freq> | Max. frequency for which the circuit model is valid | Default unit is Hz |
| <El_Length> | Electrical length (offset parameter) of the standard | Default unit is m |
| <Loss> | Loss (offset parameter) of the standard | To be specified without unit (implicit unit is dB) |
| <Z0> | Reference impedance (no unit) | To be specified without unit (implicit unit is Ω) |
| <C0> \| <L0> | Polynomial coefficients $C_0$ or $L_0$ for the fringing capacitance or residual inductance of the standard (load parameter) | To be specified without unit (implicit unit is fF or pH) |
| <C1> \| <L1> | Polynomial coefficients $C_1$ or $L_1$ for the fringing capacitance or residual inductance of the standard (load parameter) | To be specified without unit (implicit unit is fF /GHz or pH /GHz) |
| <C2> \| <L2> | Polynomial coefficients $C_2$ or $L_2$ for the fringing capacitance or residual inductance of the standard (load parameter) | To be specified without unit (implicit unit is fF/GHz$^2$ or pH/GHz$^2$) |
| <C3> \| <L3> | Polynomial coefficients $C_3$ or $L_3$ for the fringing capacitance or residual inductance of the standard (load parameter) | To be specified without unit (implicit unit is fF/GHz$^3$ or pH/GHz$^3$) |

| | | |
|---|---|---|
| OPEN \| SHORt \| MATCh | Simplified modelling as an open or short or match standard. The load circuit model generally consists of capacitance C which is connected in parallel to an inductance L and a resistance R, both connected in series. OPEN means that R is infinite so that the standard behaves like a capacitor (no inductance, the polynomial coefficients specify C). SHORt means that R is zero so that the standard behaves like an inductance (no capacitance, the polynomial coefficients specify L). MATCh means that the standard behaves like a match with the specified <Resistance> | Character data |
| <Resistance> | Optional resistance for the MATCh standard | Numeric value |
| , <Port_1>, <Port_2> | Optional port restriction: one port number for one port standards, two port numbers for two port standards | Integer value |

The different standard types are defined by the following parameters:

| <std_type> | Meaning | Parameters in `[SENSe<Ch>:]CORRection:CKIT:<conn_type>:<std_type>` |
|---|---|---|
| MOPen \| FOPen | Open (m) or open (f) | '<Ckit_Name>' ... <C3>[, <Port_1>]<br>complete parameter list with capacitance coefficients, no OPEN \| SHORT \| MATCh |
| MSHort \| FSHort | Short (m) or short (f) | '<Ckit_Name>' ... <L3>[, <Port_1>]<br>complete parameter list with inductance coefficients, no OPEN \| SHORT\| MATCh |
| OSHort \| MOSHort \| FOSHort | Offset short (sexless) or offset short (m) or offset short (f) (for user-defined connector types only) | '<Ckit_Name>' ... <L3>[, <Port_1>]<br>complete parameter list with inductance coefficients, no OPEN \| SHORT\| MATCh |
| MMTCh \| FMTCh | Match (m) or match (f) | '<Ckit_Name>' ... <Max_Freq>[, <Port_1>]<br>no offset parameters, no load parameters (polynomial coefficients), no OPEN \| SHORt\| MATCh |
| MSMatch \|FSMatch | Sliding match (m) or sliding match (f) | '<Ckit_Name>' ... <Max_Freq>[, <Port_1>]<br>no offset parameters, no load parameters (polynomial coefficients), no OPEN \| SHORt\| MATCh |
| MREFlect \| FREFlect | Reflect (m) or reflect (f) | '<Ckit_Name>' ... OPEN \| SHORt[, <Port_1>]<br>no loss, otherwise complete parameter list |
| MMTHrough \| MFTHrough \| FFTHrough | Through (m - m) or through (m - f) or through (f - f) | '<Ckit_Name>' ... <Loss>[, <Port_1>, <Port_2>]<br>no load parameters (polynomial coefficients), no OPEN \| SHORt\| MATCh |
| MMLIne1 \| MFLIne1 \| FFLIne1 (suffix 1 optional) | Line1 (m - m) or line1 (m - f) or line1 (f - f) | '<Ckit_Name>' ... <Loss>[, <Port_1>, <Port_2>]<br>no load parameters (polynomial coefficients), no OPEN \| SHORt\| MATCh |
| MMLIne2 \| MFLIne2 \| | Line1 (m - m) or line2 (m - f) or line2 (f - f), | '<Ckit_Name>' ... <Loss>[, <Port_1>, <Port_2>]<br>no load parameters (polynomial coefficients), no OPEN \| SHORt\| |

| | | |
|---|---|---|
| FFLIne2 | especially for TRL calibration | MATCh |
| MMATten \| MFATten \| FFATten | Attenuation (m - m) or attenuation (m - f) or attenuation (f - f) | '<Ckit_Name>' ... <Max_Freq>[, <Port_1>, <Port_2>]<br>no offset parameters, no load parameters (polynomial coefficients), no OPEN \| SHORt\| MATCh |
| MMSNetwork \| MFSNetwork \| FFSNetwork | Symmetric network (m - m) or symmetric network (m - f) or symmetric network (f - f) | '<Ckit_Name>' ... OPEN \| SHORt\| MATCh[, <Port_1>, <Port_2>]<br>no loss, otherwise complete parameter list |

## [SENSe<Ch>:]CORRection:CKIT:<conn_type>:SELect '<ckit_name>'

Selects the calibration kit to be used for a specified connector type <conn_type>.

For connector types with arbitrary, user-defined names you can use the command `[SENSe<Ch>:]CORRection:CKIT:SELect` '<conn_name>', '<ckit_name>'.

**<Ch>** Channel number. This suffix is ignored because calibration kits are channel-independent.

**<conn_type>** 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* ...

**'<ckit_name>'** String parameters containing the name of a calibration kit available on the analyzer.

*RST value – (A *RST does not change the assignment between connector types and calibration kits.)

**SCPI, Command Types** Device-specific, command or query

**Example:**

```
MMEM:LOAD:CKIT
'C:\Rohde&Schwarz\NWA\Calibration\Kits\New_kit.calkit'
```

```
Load the previously created cal kit file New_kit.calkit
from the default cal kit directory.
```

```
CORR:CKIT:N50:SEL 'New_kit'
```

Assign the imported kit to the N 50 Ω connector type (assuming that the cal kit name stored in `New_kit.calkit` reads New_kit).

**Cal kit naming conventions**

Calibration kit names must be entered as string parameters. The string contains the cal kit name 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*.

## [SENSe<Ch>:]CORRection:CKIT:SELect '<conn_name>', '<ckit_name>'

Selects the calibration kit to be used for a specified connector type <conn_type>.

The command is suitable for connector types with arbitrary, user-defined names. For standard connector types you can use the command `[SENSe<Ch>:]CORRection:CKIT:<conn_type>:SELect`.

| | |
|---|---|
| **<Ch>** | Channel number. This suffix is ignored because calibration kits are channel-independent. |
| **'<conn_name>'** | Connector type, e.g. a user-defined connector type (string variable) |
| **'<ckit_name>'** | String parameters containing the name of a calibration kit available on the analyzer. |
| *RST value | – (A *RST does not change the assignment between connector types and calibration kits.) |
| **SCPI, Command Types** | Device-specific, command or query (the query requires the first string parameter only) |
| **Example:** | `MMEM:LOAD:CKIT 'C:\Rohde&Schwarz\NWA\Calibration\Kits\New_kit.calkit'` |
| | `Load the previously created cal kit file New_kit.calkit from the default cal kit directory.` |
| | `CORR:CKIT:SEL 'N 50 Ohm', 'New_kit'` |
| | Assign the imported kit to the N 50 Ω connector type (assuming that the cal kit name stored in `New_kit.calkit` reads New_kit). |

## [SENSe<Ch>:]CORRection:CKIT:INSTall '<file_name>'

Loads cal kit data from a specified ZVR cal kit file.

| | |
|---|---|
| **<Ch>** | Channel number. This suffix is ignored because calibration kits are channel-independent. |
| **'<file_name>'** | String parameter to specify the name and directory of the cal kit file to be loaded.<br>*Note: The loaded file must be a ZVR-specific cal kit file with the extension *.ck. NWA cal kit files (*.calkit) can be imported using the `MMEMory:LOAD:CKIT` command. Agilent cal kit files can be imported manually and converted into *.calkit files.* |
| *RST value | – |
| **SCPI, Command Types** | Device-specific, no query. |
| **Example:** | `CORR:CKIT:INST 'C:\Rohde&Schwarz\NWA\Calibration\Kits\ZCAN.ck'` |
| | `Load the previously created ZVR cal kit file ZCAN.ck from` |

```
the default cal kit directory.

MMEM:STOR:CKIT 'ZCAN',
'C:\Rohde&Schwarz\NWA\Calibration\Kits\ZCAN.calkit'

Store the imported cal kit data to a NWA cal kit file
ZCAN.calkit (assuming that the cal kit name stored in
ZCAN.ck reads ZCAN).
```

## [SENSe<Ch>:]CORRection:COLLect...

This subsystem controls the system error correction and manages calibration kits.

## [SENSe<Ch>:]CORRection:COLLect[:ACQuire] THRough | OPEN1 | OPEN2 | OPEN12 | SHORT1 | SHORT2 | SHORT12 | MATCH1 | MATCH2 | MATCH12 | NET | ATT | IMATCH12 | REFL1 | REFL2 | SLIDE1 | SLIDE2 | SLIDE12 | LINE1 | LINE2 | M1O2 | O1M2 | M1S2 | S1M2 | OSHORT1 | OSHORT2 | UTHRough[, <Dispersion>, AUTO | <delay | phase>]

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.

Use the generalized command `[SENSe<Ch>:]CORRection:COLLect[:ACQuire]:SELected` to obtain measurement data at arbitrary analyzer ports.

| | |
|---|---|
| **<Ch>** | Channel number of the calibrated channel. If unspecified the numeric suffix is set to 1. |
| **THRough ... UTHRough** | Standard types: *Through, Open, Short, Match (MATCH12 and IMATCH12 are synonymous), Symmetric Network (NET), Attenuation (ATT), Reflect, Sliding Match (SLIDe), Line1 (LINE1 and LINE are synonymous), Line2 (esp. for TRL calibration), Match/Open (M1O2, O1M2), Match/Short (M1S2, S1M2), Offset Short (OSHort), Unknown Through (UTHRough)*.<br>The numbers in the parameters denote the analyzer ports. Two numbers 12 mean that two separate calibrations are performed at ports 1 and 2. |
| *RST value | ON |
| **<Dispersion>** | Optional status parameter for UTHRough standard:<br>OFF – Unknown through standard is non-dispersive<br>ON – Unknown through standard is dispersive |
| *RST value | OFF |
| **AUTO \| <delay \| phase>** | Optional entry of delay time or phase for UTHRough standard:<br>AUTO – The analyzer determines the delay time or phase during the calibration sweep<br><delay or phase> – 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).<br>See also background information for `[SENSe<Ch>:]CORRection:COLLect[:ACQuire]:SELected`. |
| *RST value | AUTO |

| | |
|---|---|
| **SCPI, Command Types** | Confirmed (with device-specific standards), no query |
| **Example:** | See [SENSe<Ch>:]CORRection:COLLect:SAVE. |

## [SENSe<Ch>:]CORRection:COLLect[:ACQuire]:RSAVe <Boolean>

Activates or deactivates the calibration mode where the raw measurement data of the standards is stored after the calibration is completed. The setting is valid for the current calibration, where it overwrites the global setting ([SENSe<Ch>:]CORRection:COLLect[:ACQuire]:RSAVe:DEFault). A new calibration deletes the calibration data acquired in previous calibrations.

| | |
|---|---|
| **<Ch>** | Channel number of the calibrated channel. |
| **<Boolean>** | ON \| OFF - Keep measurement data on or off. |
| *RST value | OFF |
| **SCPI, Command Types** | Device-specific, command or query |
| **Example:** | `CORR:COLL:RSAV:DEF ON` |
| | Generally keep the raw measurement data of the standards after the calibration is completed. |
| | `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 raw measurement results of this standard. |
| | `CORR:COLL:RSAV OFF` |
| | To save disk space, delete the current raw calibration data after the calibration is completed. |
| | `CORR:COLL:SAVE:SEL` |
| | Calculate the system error correction data and apply them to the active channel. |

## [SENSe<Ch>:]CORRection:COLLect[:ACQuire]:RSAVe:DEFault <Boolean>

Activates or deactivates the calibration mode where the raw measurement data of the standards is stored after the calibration is completed. The setting remains valid for all subsequent calibrations until it is changed explicitly. A new calibration deletes the calibration data acquired in previous calibrations.

| | |
|---|---|
| **<Ch>** | Channel number of the calibrated channel. |
| **<Boolean>** | ON \| OFF - Keep measurement data on or off. |
| *RST value | OFF |
| **SCPI, Command Types** | Device-specific, command or query |
| **Example:** | See [SENSe<Ch>:]CORRection:COLLect[:AQUire]:RSAVe. |

## [SENSe<Ch>:]CORRection:COLLect[:ACQuire]:SELected THRough | OPEN | SHORt | MATCh | NET | ATT | REFL | SLIDe | LINE1 | LINE2 | OSHort | UTHRough,<port_no>,<port_no>[, <Dispersion>, AUTO | <delay | phase>]

Starts a calibration measurement in order to acquire measurement data for the selected standards. The standards are reflection or transmission standards and can be connected to arbitrary analyzer ports.

| | |
|---|---|
| **<Ch>** | Channel number of the calibrated channel. |
| **THRough ... UTHRough** | Standard types: *Through, Open, Short, Match, Symmetric Network (NET), Attenuation (ATT), Reflect, Sliding Match (SLIDe), Line1 (LINE1 and LINE are synonymous), Line2 (esp. for TRL calibration), Offset Short, Unknown Through*. |
| **<port_no>** | Port numbers of the analyzer. For a transmission standard (through, line, attenuation, symmetric network) the input and output port numbers must be specified, for reflection standards, only one port number is required. |
| *RST value | – |
| **<Dispersion>** | Optional status parameter for UTHRough standard:<br>OFF – Unknown through standard is non-dispersive<br>ON – Unknown through standard is dispersive |
| ***RST value** | OFF |
| **AUTO \| <delay \| phase>** | Optional entry of delay time or phase for UTHRough standard:<br>AUTO – The analyzer determines the delay time or phase during the calibration sweep<br><delay or phase> – entry of the delay time in ps (for non-dispersive standards) or of the 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.<br> Automatic **determination of the phase**<br>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.<br>In remote control, the analyzer performs a plausibility check in order 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. |
| ***RST value** | AUTO |
| **SCPI, Command Types** | Device-specific, no query |
| **Example:** | `See [SENSe<Ch>:]CORRection:COLLect:SAVE:SELected.` |

## [SENSe<Ch>:]CORRection:COLLect:AUTO '<file_name>',<port_no>{,<port_no>}

Selects and initiates a one-port, two-port, three-port or four-port automatic calibration at arbitrary

analyzer and cal unit ports.

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`.

If several calibration units are connected, use `SYSTem:COMMunicate:RDEVice:AKAL:ADDRess` to select a unit for the calibration.

**<Ch>** Channel number of the calibrated channel.

**'<file_name>'** Name and (possibly) directory of the cal kit file to be used for the automatic calibration (string parameter):

- If nothing is specified (empty string ' '), the default cal kit file stored in the calibration unit is used. This file is also used in manual control.
- A NWA cal kit file name *.calkit without path denotes a specific cal kit file stored in the Calibration Unit.
- A NWA cal kit file name *.calkit with path denotes a specific cal kit file stored in an arbitrary directory.

**<port_no>** Port numbers of the analyzer. For an n-port automatic calibration (n = 1 to 4), n arbitrary (not necessarily consecutive) port numbers must be specified. The analyzer automatically detects the calibration unit ports connected to each analyzer port.

*RST value –

**SCPI, Command Types** Device-specific, no query

**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.

## [SENSe<Ch>:]CORRection:COLLect:AUTO:CKIT '<file_name>'

Generate a 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 stored in the calibration unit or written to a directory on the analyzer.

**<Ch>** Channel number of the calibrated channel (not relevant for this command).

**'<file_name>'** Name and (possibly) directory of the created cal kit file (string parameter, extension *.calkit):

- If no path and directory is specified, the file is stored in the calibration unit (see also `[SENSe<Ch>:]CORRection:COLLect:AUTO`). The factory calibration data on the unit is not overwritten.
- If a path and directory is specified, the file is stored in the directory on the analyzer's hard disk.

The default directory (`MMEMory:CDIRectory`) is not used.

*RST value –

**SCPI, Command Types** Device-specific, no query

**Example:**

```
CORR:COLL:AUTO:CKIT 'AutoCalChar.calkit'
```

Generate a cal kit file AutoCalChar.calkit for the active calibration unit and store it internally.

## [SENSe<Ch>:]CORRection:COLLect:AUTO:PORTs '<file_name>',<analyzer_port_no>,<cal_unit_port_no>{,<analyzer_port_no>,<cal_unit_port_no>}

Selects and initiates a one-port, two-port, three-port or four-port automatic calibration at arbitrary analyzer and calibration unit ports.

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`.

**<Ch>** Channel number of the calibrated channel.

**'<file_name>'** Name and (possibly) directory of the cal kit file to be used for the automatic calibration (string parameter):

- If nothing is specified (empty string ' '), the default cal kit file stored in the calibration unit is used. This file should be used to operate the calibration unit like in manual control; specifying another cal kit file is primarily for service purposes.
- A NWA cal kit file name *.calkit without path denotes a specific cal kit file stored in the Calibration Unit.
- A NWA cal kit file name *.calkit with path denotes a specific cal kit file stored in an arbitrary directory.

**<analyzer_port_no>** Port numbers of the analyzer. For an n-port automatic calibration (n = 1 to 4), n arbitrary (not necessarily consecutive) port numbers must be specified.

**<cal_unit__port_no>** Port numbers of the cal unit. For an n-port automatic calibration (n = 1 to 4), 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.

*RST value –

**SCPI, Command Types** Device-specific, no query

**Example:**

```
CORR:COLL:AUTO:PORT '', 1, 2, 2, 4, 4, 1
```

Perform an automatic 3-port calibration at the analyzer 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.

## [SENSe<Ch>:]CORRection:COLLect:AUTO:PORTs:CONNection?

Returns the assignment between the network analyzer ports and the ports of the connected automatic

calibration unit.

**<Ch>** Channel number of the calibrated channel (not relevant for this command).

*RST value –

**SCPI, Command Types** Device-specific, query only

**Example:**

```
CORR:COLL:AUTO:PORT '', 1, 2, 2, 4, 4, 1
```

Perform an automatic 3-port calibration at the analyzer 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.

```
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.

## [SENSe<Ch>:]CORRection:COLLect:CONNection<port_no> N50FEMALE | N50MALE | N75FEMALE | N75MALE | PC7 | PC35FEMALE | PC35MALE | PC292FEMALE | PC292MALE

## Additional parameters (): UFEMALE1 | UMALE1 | UFEMALE2 | UMALE2 | SMAFEMALE | SMAMALE

Selects a connector type at a specified port <port_no> and its gender.

Use `[SENSe<Ch>:]CORRection:COLLect:SCONnection<port_no>` to select an arbitrary connector type using a string variable.

**<Ch>** Channel number of the calibrated channel.

**<port_no>** Port numbers of the analyzer.

***Note:*** *If the analyzer is set to use the same connectors at all ports (`[SENSe<Ch>:]CORRection:COLLect:CONNection<port_no>:PORTs ALL`), then a change of a connector type is valid for all ports. The gender of the connectors can still be different.*

**Parameters** Connector type and gender of the connectors (omitted for query). UFEMALE1 and UMALE1 denote the user-defined connector type *UserConn1,* UFEMALE2 and UMALE2 denote the user-defined connector type *UserConn2.* SMAFEMALE and SMAMALE denote the user-defined connector type *SMA.* The user-defined connector types must be defined before being addressed by `[SENSe<Ch>:]CORRection:COLLect:CONNection<port_no>`.

*RST value N50FEMALE for all ports.

**SCPI, Command Types** Device-specific, command or query.

**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`.

## [SENSe<Ch>:]CORRection:COLLect:CONNection:PORTs ALL | SINGle

Qualifies whether the connector types at the analyzer ports (but not their gender) are equal or independent.

*In the present firmware version, a **calibration** must be performed using the same connector type at all ports. Use [SENSe<Ch>:]CORRection:COLLect:CONNection<port_no>:PORTs SINGle if you want to perform a **measurement** with independent port connectors.*

**<Ch>** Channel number of the calibrated channel.

**Parameters** 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.

**<port_no>** Port numbers of the analyzer. This parameter has no effect because the setting affects all ports.

*RST value ALL

**SCPI, Command Types** Device-specific, command or query.

**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.

## [SENSe<Ch>:]CORRection:COLLect:DELete ['<cal_name>']

Deletes system error correction data generated and stored previously.

**<Ch>** Channel number of the calibrated channel.

**'<cal_name>'** Name of the calibration (string parameter) defined together with calibration type `[SENSe<Ch>:]CORRection:COLLect:METHod:DEFine`. If nothing is specified the analyzer deletes the last system error correction stored by means of `[SENSe<Ch>:]CORRection:COLLect:SAVE`.

*RST value –

**SCPI, Command Types** Device-specific, no query

**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; DEL
```

Calculate the system error correction data and apply them to the active channel, then delete the data.

## [SENSe<Ch>:]CORRection:COLLect:METHod FRTRans | TOM | TRM | TRL | TNA | TOSM | FOPort1 | FOPort2 | FOPort12 | FOPTport | ROPTport | REFL1 | REFL2 | REFL12 | TPORt | UOSM

Selects a one-port or two-port calibration type at ports 1/2.

Use the generalized command `[SENSe<Ch>:]CORRection:COLLect:METHod:DEFine` to select the calibration type for arbitrary analyzer ports or a multiport calibration type.

| | |
|---|---|
| **<Ch>** | Channel number of the calibrated channel. |
| **Parameters** | Calibration types: *TOM, TRM, TRL, TNA, TOSM, Full One Port, One Path Two Port, Normalization* (REFL1, REFL2 and REFL12 for one-port, TPORT for two-port)*, TOSM with unknown through*.<br>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. |
| *RST value | – |
| **SCPI, Command Types** | Confirmed (with device-specific calibration types), command or query. If several calibration types are assigned to channel <Ch>, the query returns a list of all calibration types. |
| **Example:** | See [SENSe<Ch>:]`CORRection:COLLect:SAVE`. |

## [SENSe<Ch>:]CORRection:COLLect:METHod:DEFine '<cal_name>', REFL | RSHort | FOPort | FRTRans | OPTPort | TOSM | TOM | TRM | TRL | TNA | UOSM, <port_no>[,<port_no>][,<port_no>][,<port_no>]

Selects a one-port, two-port, three-port or four-port calibration type at arbitrary analyzer ports.

| | |
|---|---|
| **<Ch>** | Channel number of the calibrated channel. |
| **'<cal_name>'** | 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`). |
| **Parameters** | Calibration types: *One-port Normalization (reflection), using an open* (REFL) *or a short (RSHort) standard, Full One Port, Two-port Normalization (transmission,* FRTRans)*, One Path Two Port, TOSM*, *TOM, TRM, TRL, TNA, TOSM with unknown through*. |
| **<port_no>** | Port numbers of the analyzer. For an n-port calibration type (n = 1 to 4), 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. |
| *RST value | – |
| **SCPI, Command Types** | Device-specific, no query. Use `[SENSe<Ch>:]CORRection:COLLect:METHod?` to obtain a list of all calibration types for channel <Ch>. |
| **Example:** | `See [SENSe<Ch>:]CORRection:COLLect:SAVE:SELected.` |

## [SENSe<Ch>:]CORRection:COLLect:SAVE

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 <Ch>, '<file_name>'`).

This command is the ZVR-compatible equivalent of `[SENSe<Ch>:]CORRection:COLLect:SAVE:SELected`. It must be used in combination with the ZVR-compatible commands for calibration method and standard selection; see example below.

| | |
|---|---|
| **<Ch>** | Channel number of the calibrated channel. |
| *RST value | – |
| **SCPI, Command Types** | Confirmed, no query |
| **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 them to the active channel. |

## [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.

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.

This command must be used in combination with the 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 ZVA-specific calibration commands or if you want to calibrate more than 2 ports.

| | |
|---|---|
| **<Ch>** | Channel number of the calibrated channel. |
| *RST value | – |
| **SCPI,** | Device-specific, no query |

**Command Types**

**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.

## [SENSe<Ch>:]CORRection:COLLect:SAVE:SELected

Calculates the system error correction data from the acquired measurement results (`[SENSe<Ch>:]CORRection:COLLect[:ACQuire]:SELected`), 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 <Ch>, '<file_name>'`).

**<Ch>** Channel number of the calibrated channel.

*RST value –

**SCPI, Command Types** Confirmed, no query

**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
```

Calculate the system error correction data and apply them to the active channel.

## [SENSe<Ch>:]CORRection:COLLect:SAVE:SELected: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]:SELected`) is taken into account.

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.

**<Ch>** Channel number of the calibrated channel.

*RST value –

**SCPI, Command Types** Device-specific, no query

**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.

## [SENSe<Ch>:]CORRection:COLLect:SCONnection<port_no> '<conn_name>', MALE | FEMale

Selects a connector type at a specified port <port_no> and its gender. In contrast to `[SENSe<Ch>:]CORRection:COLLect:CONnection<port_no>`, this command uses a string variable to identify the connector type.

**<Ch>** Channel number of the calibrated channel.

**<port_no>** Port numbers of the analyzer.

***Note:*** *If the analyzer is set to use the same connectors at all ports (`[SENSe<Ch>:]CORRection:COLLect:CONNection<port_no>:PORT`*

*s ALL), then a change of a connector type is valid for all ports. The gender of the connectors can still be different.*

**Parameters** Connector type (string parameter) and gender of the connectors. The gender designation MALE | FEMale is not necessary (and ignored) for sexless connector types.

*RST value 'N 50 Ohm',FEM for all ports.

**SCPI, Command Types** Device-specific, command or query.

**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.

## [SENSe<Ch>:]CORRection:CONNection '<conn_name>', TEM | WGUide, GENDer | NGENder, <perm_rel>, <imped>

Configures the user-defined connector types.

**<Ch>** Channel number

**'<conn_name>'** Name of the user-defined connectors, string parameter

**TEM | WGUide** Transverse electric or waveguide type propagation mode

**GENDer | NGENder** Polar connector type (m/f)
Sexless connector type

**<perm_rel>** Relative permittivity

Range [def. unit] 0.0000000001 to 1000. [–] UP/DOWN/MIN/MAX parameters are not available for this command

*RST value –

**<imped>** For TEM type connectors: reference impedance in Ω (without unit),
For WGUide type connectors: cutoff frequency in Hz (without unit)

Range [def. unit] Ref. impedance: 1μΩ to 1000 MΩ
Cutoff frequency: 0 Hz to 1000 GHz

*RST value –

**SCPI, Command Types** Device-specific, command or query

**Example:**

```
CORR:CONN 'USERCON',TEM,GEND,1.00000,50
```

Define a TEM type connector type names USERCON.

```
CORR:CONN? 'USERCON'
```

Query the properties of the configured connector type.

```
CORR:CONN:DEL 'USERCON'
```

Delete the configured connector type.

## [SENSe<Ch>:]CORRection:CONNection:DELete '<conn_name>'

Deletes a user-defined connector type named '<conn_name>'.

| | |
|---|---|
| **<Ch>** | Channel number |
| **'<conn_name>'** | Name of the user-defined connectors, string parameter |
| **SCPI, Command Types** | Device-specific, no query |
| **Example:** | `See [SENSe<Ch>:]CORRection:CONNection` |

## [SENSe<Ch>:]CORRection:DATA 'SCORR1' | ... | 'SCORR12' | 'G11' | ... 'G22' | 'H11' | ... | H22'

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`.*

Use the generalized command `[SENSe<Ch>:]CORRection:COLLect:CDATa` to transfer calibration data for arbitrary analyzer ports.

| | |
|---|---|
| **<Ch>** | Channel number of the calibrated channel. If unspecified the numeric suffix is set to 1. |
| **Error term parameters** | 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. |
| Range [def. unit] | The error terms consist of dimensionless complex numbers. The parameters must be transferred in full length and have the following meaning:<br>'SCORR1' – Directivity at port 1<br>'SCORR2' – Source match at port 1<br>'SCORR3' – Reflection tracking at port 1<br>'SCORR4' – Forward isolation between port 1 and port 2<br>'SCORR5' – Load match at port 2<br>'SCORR6' – Forward transmission tracking between port 1 and port 2<br>'SCORR7' – Directivity at port 2<br>'SCORR8' – Source match at port 2<br>'SCORR9' – Reflection tracking at port 2<br>'SCORR10' – Reverse isolation between port 2 and port 1<br>'SCORR11' – Load match at port 1<br>'SCORR12' – Reverse transmission tracking between port 2 and port 1<br>'G11' ... 'G22' – G matrix elements<br>'H11' ... 'H22' – H matrix elements<br>**G and H Matrices**<br>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 $H_{21} = 1$.

*RST value –
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`.

**SCPI, Command Types** Device-specific, command or query

NOTE *After a system error correction in one of the sweep types Power, Time, or CW Mode, the query returns two sets of correction data, corresponding to the fixed CW frequency and the CW frequency plus 1 Hz (minus 1 Hz if the CW frequency is equal to the maximum frequency of the analyzer).*

**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<br>REFL2<br>REFL12 | 'SCORR3'<br>'SCORR9'<br>'SCORR3' and 'SCORR9' |
| Full one port | FOPort1<br>FOPort2<br>FOPort12 | 'SCORR1' to 'SCORR3'<br>'SCORR7' to 'SCORR9'<br>'SCORR1' to 'SCORR3' and 'SCORR7' to 'SCORR9' |
| Two-port normalization | FTRans<br>RTRans<br>FRTRans | 'SCORR6'<br>'SCORR12'<br>'SCORR6' and 'SCORR12' |
| One path two port | FOPTport<br>ROPTport | 'SCORR1' to 'SCORR3', 'SCORR6'<br>'SCORR7' to 'SCORR9', 'SCORR12' |

| | | |
|---|---|---|
| TOSM | TOSM | 'SCORR1' to 'SCORR12' (at present the isolation terms 'SCORR4' and 'SCORR10' are not included) |
| TOM, TRM, TRL, TNA | TOM \| TRM \| TRL \| TNA | 'DIRECTIVITY', 'SRCMATCH', 'REFLTRACK', 'LOADMATCH', 'TRANSTRACK' (for reading and writing)<br>'G11' ... 'G22' and 'H11', 'H12', 'H22' (for reading only; the 'H21' matrix elements are normalized to 1) |

## [SENSe<Ch>:]CORRection:DATA:PARameter?

Returns the sweep settings of the active system error correction for channel <Ch>.

**<Ch>** Channel number of the calibrated channel.

**Response** Five values, corresponding to the start and stop frequency of the calibration sweep, the number of points, the source power, and the sweep type (LIN | LOG | SEGM).

*RST value – (the command generates an execution error if no system error correction is active)

**SCPI, Command Types** Device-specific, query only

**Example:**

```
See [SENSe<Ch>:]CORRection:DATE?
```

## [SENSe<Ch>:]CORRection:DATE?

Returns the date and time when the active system error correction data for channel <Ch> was acquired.

**<Ch>** Channel number of the calibrated channel.

*RST value – (the command generates an execution error if no system error correction is active)

**SCPI, Command Types** Device-specific, query only

**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 them to the active channel.

```
CORR:DATE?
```

Query the time when the system error correction became active. The analyzer returns the data and time, e.g. *'03/20/06,18:30:39'*.

```
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 $S_{21}$).

## [SENSe<Ch>:]CORRection:EDELay<port_no>:AUTO ONCE

Defines the offset parameter for the active test port such that the residual delay of the active trace (defined as the negative derivative of the phase response) is minimized across the entire sweep range.

| | |
|---|---|
| **<Ch>** | Channel number of the offset-corrected channel. |
| **<port_no>** | Port number of the analyzer. This numeric suffix is ignored; the active port is determined by the active trace. |
| **ONCE** | Applies the *Auto Length* function. |
| *RST value | – |
| **SCPI, Command Types** | Device-specific, no query |
| **Example:** | `*RST; CORR:EDEL:AUTO ONCE` |
| | Reset the instrument and apply the auto length function to the default trace *(Trc1* in channel 1). |

## [SENSe<Ch>:]CORRection:EDELay<port_no>:DIELectric <permittivity>

Defines the permittivity for the offset correction at test port <port_no>.

| | |
|---|---|
| **<Ch>** | Channel number of the offset-corrected channel |
| **<port_no>** | Port number of the analyzer |
| **<permittivity>** | Permittivity |
| Range [def. unit] | 1 to +3.4028234664E+038 m |
| *RST value | 1.00062 [–] |
| **SCPI, Command Types** | Device-specific, command or query |
| **Example:** | `See [SENSe<Ch>:]CORRection:EDELay<port_no>:ELENgth` |

## [SENSe<Ch>:]CORRection:EDELay<port_no>:DISTance <length>

Defines the offset parameter for test port <port_no> as a mechanical length.

| | |
|---|---|
| **<Ch>** | Channel number of the offset-corrected channel |
| **<port_no>** | Port number of the analyzer |
| **<length>** | Mechanical length |
| Range [def. unit] | –3.402823466E+038 m to +3.4028234664E+038 m. In contrast to the electrical length ([SENSe<Ch>:]CORRection:EDELay<port_no>:ELENgth), the distance can |

| | |
|---|---|
| | not be incremented (the UP/DOWN parameters do not work). |
| *RST value | 0 m [m] |
| **SCPI, Command Types** | Device-specific, command or query |
| **Example:** | `See [SENSe<Ch>:]CORRection:EDELay<port_no>:ELENgth` |

## [SENSe<Ch>:]CORRection:EDELay<port_no>:ELENgth <length>

Defines the offset parameter for test port <port_no> as an electrical length.

| | |
|---|---|
| **<Ch>** | Channel number of the offset-corrected channel |
| **<port_no>** | Port number of the analyzer |
| **<length>** | Electrical length |
| Range [def. unit] | –3.4028234664E+038 m to +3.4028234664E+038 m [m]. The increment (UP, DOWN) is 1 cm. |
| *RST value | 0 m |
| **SCPI, Command Types** | Device-specific, command or query |
| **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. |

## [SENSe<Ch>:]CORRection:EDELay<port_no>[:TIME] <delay>

Defines the offset parameter for test port <port_no> as a delay time.

| | |
|---|---|
| **<Ch>** | Channel number of the offset-corrected channel |
| **<port_no>** | Port number of the analyzer |
| **<delay>** | Delay |
| Range [def. unit] | -3.40282346638529E+038 s to +3.40282346638529E+038 s [s]. In contrast to the electrical length ([SENSe<Ch>:]CORRection:EDELay<port_no>:ELENgth), the delay time can not be incremented (the UP/DOWN parameters do not work). |
| *RST value | 0 s |
| **SCPI,** | Device-specific, command or query |

**Command Types**

**Example:** See [SENSe<Ch>:]CORRection:EDELay<port_no>:ELENgth

## [SENSe<Ch>:]CORRection:FACTory[:STATe] <Boolean>

Enables or disables the factory calibration for a particular channel.

**<Ch>** Channel number of the calibrated channel

**<Boolean>** Enable or disable factory calibration
*RST value ON

**SCPI, Command Types** Device-specific, command or query

**Example:**

```
*RST; CORR:FACT?
```

Reset the instrument and query whether the factory calibration for channel 1 is enabled. The response is *1*.

## [SENSe<Ch>:]CORRection:LOSS<port_no> <DC_loss>

Defines the frequency-independent part (DC value) of the offset loss.

**<Ch>** Channel number of the offset-corrected channel

**<port_no>** Port number of the analyzer

**<DC-loss>** Frequency-independent part of the offset loss
Range [def. unit] -200 dB to +200 dB [dB]. The increment (UP/DOWN) is 0.001 dB.
*RST value 0 dB

**SCPI, Command Types** Confirmed, command or query

**Example:** See [SENSe<Ch>:]CORRection:EDELay<port_no>:ELENgth

## [SENSe<Ch>:]CORRection:LOSS<port_no>:AUTO ONCE

Defines the offset parameters for the active test port such that the residual delay of the active trace (defined as the negative derivative of the phase response) is minimized and the measured loss is reproduced as far as possible across the entire sweep range.

**<Ch>** Channel number of the offset-corrected channel.

**<port_no>** Port number of the analyzer. This numeric suffix is ignored; the active port is determined by the active trace.

**ONCE** Applies the *Auto Length and Loss* function.
*RST value –

**SCPI, Command Types** Device-specific, no query

**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).

## [SENSe<Ch>:]CORRection:LOSS<port_no>:FREQuency <ref_frequency>

Defines the reference frequency for the frequency-dependent part of the offset loss (`[SENSe<Ch>:]CORRection:LOSS<port_no>:OFFSet`).

| | |
|---|---|
| **<Ch>** | Channel number of the offset-corrected channel |
| **<port_no>** | Port number of the analyzer |
| **<ref_frequency>** | Reference frequency |
| Range [def. unit] | Frequency range of the analyzer model [Hz]. The increment (UP/DOWN) is 1 MHz. |
| *RST value | 1000000000 Hz (= 1 GHz) |
| **SCPI, Command Types** | Device-specific, command or query |
| **Example:** | `See [SENSe<Ch>:]CORRection:EDELay<port_no>:ELENgth` |

## [SENSe<Ch>:]CORRection:LOSS<port_no>:OFFSet <ref_loss>

Defines the offset loss at the reference frequency (`[SENSe<Ch>:]CORRection:LOSS<port_no>:FREQuency`).

| | |
|---|---|
| **<Ch>** | Channel number of the offset-corrected channel |
| **<port_no>** | Port number of the analyzer |
| **<ref_loss>** | Frequency-dependent part of the offset loss |
| Range [def. unit] | -200 dB to +200 dB [dB]. The increment (UP/DOWN) is 0.001 dB. |
| *RST value | 0 dB |
| **SCPI, Command Types** | Device-specific, command or query |
| **Example:** | `See [SENSe<Ch>:]CORRection:EDELay<port_no>:ELENgth` |

## [SENSe<Ch>:]CORRection:OFFSet<port_no>:MAGNitude <ref_loss>

Defines the offset loss at the reference frequency (`[SENSe<Ch>:]CORRection:LOSS<port_no>:FREQuency`).

Use the `[SENSe<Ch>:]CORRection:LOSS<port_no>...` commands to define the complete set of loss offset parameters. `[SENSe<Ch>:]CORRection:OFFSet<port_no>:MAGNitude` is equivalent to `[SENSe<Ch>:]CORRection:LOSS<port_no>:OFFSet`

| | |
|---|---|
| **<Ch>** | Channel number of the offset-corrected channel |
| **<port_no>** | Port number of the analyzer |
| **<ref_loss>** | Frequency-dependent part of the offset loss |
| Range [def. unit] | -200 dB to +200 dB [dB]. The increment (UP/DOWN) is 0.001 dB. |
| *RST value | 0 dB |

**SCPI, Command Types** Confirmed, command or query

## [SENSe<Ch>:]CORRection:OFFSet<port_no>[:STATe] <Boolean>

Resets the offset parameters for all test ports to zero or queries whether any of the offset parameters are different from zero.

| | |
|---|---|
| **<Ch>** | Channel number of the offset-corrected channel |
| **<port_no>** | Port number of the analyzer. This numeric suffix is ignored; the command affects the parameters of all ports. |
| **<Boolean>** | The parameter function depends on whether the command is used as a setting command or as a query:<br>**For setting command:**<br>ON – no effect<br>OFF – resets all length offsets to zero<br>**For query:**<br>1 – at least one length offset is different from zero<br>0 – all length offsets are zero |
| *RST value | OFF |
| **SCPI, Command Types** | Device-specific, command or query |
| **Example:** | `*RST; CORR:OFFS?`<br>Reset the instrument and query whether the length offset parameters have been reset as well. The response is *0*. |

## [SENSe<Ch>:]CORRection:POWer<port_no>...

This subsystem controls the receiver power calibration.

## [SENSe<Ch>:]CORRection:POWer<port_no>[:ACQuire] AWAVe | BWAVe | B1 | B2 | B3 | B4[,<cal_port>, PORT | GENerator, <source_no>]

Selects the wave quantity and the source for the receiver power calibration, starts the calibration sweep, and applies the receiver power correction.

NOTE *The command triggers an event and has no query form. The calibrated waves and the used source are not shown in the Receiver Power Cal dialog.*

| | |
|---|---|
| **<Ch>** | Calibrated channel number |
| **<port_no>** | Calibrated port number. This suffix is ignored because the port number is specified in the parameter list. |
| **AWAVe** | Calibration of reference wave $a_1$, $a_2$, ... |
| **BWAVe** | Calibration of received wave $b_1$, $b_2$, ....<br>The port number <cal_port>, the used source (PORT \| GENerator), and the source number <source_no> must be specified in addition. Alternative: Parameters B1 and B2. |
| **B1 \| B2 \| B3 \|** | Direct wave and cal port setting for received waves $b_1$, $b_2$,... The parameters B1 |

| | |
|---|---|
| **B4** | and B2 are ZVR-compatible. No additional parameters need to be specified. The source for B<no> is Port <no>. |
| **<cal_port>** | Calibrated port number |
| Range [def. unit] | 1 to port number of the analyzer [–] |
| **PORT \|** | Internal source at port <source_no>, to be fed to port <cal_port> using an external through connection. |
| **GENerator** | External generator no. <source_no> |
| **<source_no>** | Number of the port for the internal source or of the generator |
| Range [def. unit] | 1 to port number of the analyzer/number of external generators [–] |
| *RST values | – (for all parameters) |
| **SCPI, Command Types** | Device-specific, no query |

**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:POWer<port_no>[:STATe]`.

## [SENSe<Ch>:]CORRection:POWer<port_no>:AWAVe[:STATe] <Boolean>

Enables or disables the receiver power calibration for channel <Ch> and for the reference waves $a_n$. The command is disabled unless the reference waves have been power calibrated (`[SENSe<Ch>:]CORRection:POWer<port_no>:ACQuire AWAVe,...`).

| | |
|---|---|
| **<Ch>** | Calibrated channel number |
| **<port_no>** | Calibrated port number |
| **<Boolean>** | Enables (ON) or disables (OFF) the receiver power calibration for the reference waves $a_n$. |
| *RST value | OFF [–] |
| **SCPI, Command Types** | Device-specific, command or query |

**Example:**

```
See [SENSe<Ch>:]CORRection:POWer<port_no>:ACQuire
```

## [SENSe<Ch>:]CORRection:POWer<port_no>:DATA '<wave_string>', <block> | <numeric_value>...

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.

| | |
|---|---|
| **<Ch>** | Calibrated channel number |
| **<port_no>** | Calibrated port number |
| **'<wave_string>'** | Identifier for the calibrated wave:<br>• 'A<n>' or 'AWAV<n>' denote correction data for the reference wave a<n>, where <n> corresponds to the port number.<br>• 'B<n>' or 'BWAV<n>' denote correction data for the reference wave b<n>. |
| **<block> \| <numeric_value>...** | Power correction values in block data format or as a comma-separated list of real numbers. The query returns a list of numbers. |
| Range [def. unit] | All numbers are interpreted as dB-values. |
| *RST value | – |
| **SCPI, Command Types** | Device-specific, command or query |
| **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 'AWAV2', 1, 2, 3, 4, 5, -6, -7, -8, -9, -0` |
| | Replace the correction values by ten (new) numbers. |

## [SENSe<Ch>:]CORRection:POWer<port_no>:HARMonic:ACQuire

Starts the receiver calibration (2nd power calibration step for harmonic measurements), stores and applies the calibration data.

*The IF receiver calibration relies on the source power calibration acquired in step no. 1.* *`[SENSe<Ch>:]CORRection:POWer<port_no>:HARMonic:ACQuire`* *is enabled only after step 1 has been carried out*

*(SOURce<Ch>:POWer<port_no>:CORRection:HARMonic[:ACQuire]).*

| | |
|---|---|
| **<Ch>** | Calibrated channel number |
| **<port_no>** | Port number. This suffix is ignored; the analyzer calibrates the port that is selected as a receive port for the harmonic measurement ([SENSe<Ch>:]FREQuency:CONVersion:HARMonic:RPORt). |
| **SCPI, Command Types** | Device-specific, no query |
| **Example:** | See SOURce<Ch>:POWer<port_no>:CORRection:HARMonic[:ACQuire]. |

## [SENSe<Ch>:]CORRection:POWer<port_no>:MIXer:IF:ACQuire

Starts the IF receiver calibration (2nd power calibration step for mixer measurements), stores and applies the calibration data.

*The IF receiver calibration relies on the IF source power calibration acquired in step no. 1. [SENSe<Ch>:]CORRection:POWer<port_no>:MIXer:IF:ACQuire is enabled only after step 1 has been carried out (SOURce<Ch>:POWer<Pt>:CORRection:MIXer:RF[:ACQuire]).*

| | |
|---|---|
| **<Ch>** | Calibrated channel number |
| **<port_no>** | Calibrated port number. This suffix is ignored because the IF signal is always assigned to port 2. |
| **SCPI, Command Types** | Device-specific, no query |
| **Example:** | See SOURce<Ch>:POWer<Pt>:CORRection:MIXer:RF[:ACQuire]. |

## [SENSe<Ch>:]CORRection:POWer<port_no>:STATe <Boolean>

Enables or disables the receiver power calibration for channel <Ch> and for the received waves $b_n$. The command is disabled unless the received waves have been power calibrated ([SENSe<Ch>:]CORRection:POWer<port_no>:ACQuire BWAVe,...).

| | |
|---|---|
| **<Ch>** | Calibrated channel number |
| **<port_no>** | Calibrated port number |
| **<Boolean>** | Enables (ON) or disables (OFF) the receiver power calibration for the received waves $b_n$. |
| *RST value | OFF [–] |
| **SCPI, Command Types** | Device-specific, command or query |
| **Example:** | *RST; CORR:POW:ACQ BWAVe,1,PORT,2 |

Perform a receiver power calibration of the wave $b_1$ 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<port_no>:ACQuire.`

## [SENSe<Ch>:]CORRection:SSTate?

Returns the state of the active system error correction for channel <Ch>.

| | |
|---|---|
| **<Ch>** | Channel number of the calibrated channel. |
| **Response** | Sting variable containing the calibration state label in the trace list ('Cal', 'Cai', 'Ca?', 'Cav', 'Cal Off'). |
| *RST value | '' (empty string) |
| **SCPI, Command Types** | Device-specific, query only |
| **Example:** | `See [SENSe<Ch>:]CORRection:DATE?` |

## [SENSe<Ch>:]CORRection[:STATe] <Boolean>

Enables or disables the system error correction for channel <Ch>.

| | |
|---|---|
| **<Ch>** | Calibrated channel number |
| **<Boolean>** | Enables (ON) or disables (OFF) the correction. |
| *RST value | ON [–] |
| **SCPI, Command Types** | Confirmed, command or query |
| **Example:** | `*RST; CORR?` |
| | Reset the instrument and query whether channel 1 is system error corrected. The response is *1*. |

## [SENSe<Ch>:]COUPle...

This subsystem selects the sweep type and the measurement parameter.

## [SENSe<Ch>:]COUPle ALL | NONE

Activates normal or alternate sweep mode.

| | |
|---|---|
| **<Ch>** | Channel number. If unspecified, this parameter is set to 1. |
| **Parameters** | ALL – normal sweep mode (alternate off), complete all partial measurements before proceeding to the next sweep point<br>NONE – alternate sweep mode on, reverse the order of partial measurements and sweeps |
| *RST value | ALL |
| **SCPI, Command Types** | Device-specific, command or query |
| **Example:** | `COUP NONE` |
| | Activate the alternate sweep 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. |

## [SENSe<Ch>:]FREQuency...

This subsystem sets frequency-related parameters, especially the measurement and display ranges for the different sweep types, and controls the frequency-converting measurement modes (harmonic distortion, mixer mode, etc.).

The frequency ranges for the different instrument models are listed below:

| | ZVA8 | ZVA24 | ZVA40 |
|---|---|---|---|
| **Start, Stop** | 300 kHz to 8 GHz | 10 MHz to 25 GHz<br>(the specified frequency range is 10 MHz to 24 GHz; see data sheet) | 10 MHz to 40 GHz |
| **Center** | > 300 kHz to < 8 GHz | > 10 MHz to < 25 GHz | > 10 MHz to < 40 GHz |
| **Span** | 10 mHz to 7.9997 GHz | 10 mHz to 24.99 GHz | 10 mHz to 39.99 GHz |

***Note:*** *For a frequency sweep the range can be defined alternatively by a combination of Start/Stop frequencies or a Center frequency and Span. In fixed frequency mode (power sweep or Time/CW Mode; see [SENSe<Chn>:]FUNCtion[:ON]) the measurement frequency is set by means of [SENSe<Ch>:]FREQuency:CW|FIXed.*

## [SENSe<Ch>:]FREQuency:CENTer <center_frequency>

Defines the center of the measurement and display range for a frequency sweep.

| | |
|---|---|
| **<Ch>** | Channel number. If unspecified the numeric suffix is set to 1. |
| **<center_frequency>** | Center frequency of the sweep. |
| Range [def. unit] | Depending on the instrument model [Hz]. The increment (parameters UP or DOWN) is 0.1 kHz. |
| *RST value | Center of the analyzer's maximum frequency range: $(f_{MIN} + f_{MAX})/2$ |
| **SCPI, Command Types** | Confirmed, command or query. |
| **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.

```
FREQ:CENT 100MHz
```

Set center frequency to 100 MHz.

```
FREQ:SPAN 50000
```

Set frequency span to 50 kHz.

*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 (Center – $f_{MIN}$, $f_{MAX}$ – Center).*

## [SENSe<Ch>:]FREQuency:CONVersion FUNDamental | HARMonic | SHARmonic | THARmonic | MIXer

Enables the measurement of a harmonic or the fundamental signal for channel <Ch>.

*Enabling a harmonics measurement also resets the measured quantity; see program example below.*

**<Ch>** Channel number.

**FUNDamental** Measurement of the fundamental signal (no frequency-converting mode).

**HARMonic** Harmonics measurement. The order of the measured harmonic, the source port, and the receive port must be selected using the `[SENSe<Ch>:]FREQuency:CONVersion:HARMonic...` commands; see example below.

**SHARmonic | THARmonic** Measurement of the 2nd or 3rd harmonic with source port 1 and receive port 2. These parameters have been introduced for compatibilty with the ZVR. While they are active, a query returns `HARM`.

**MIXer** Scalar mixer mode. The mixer mode is configured using the `[SENSe<Ch>:]FREQuency:CONVersion:MIXer...` commands.

*RST value FUNDamental

**SCPI, Command Types** Device-specific, command or query. Returned values: `FUND` or `HARM`.

**Example:**

```
*RST; FREQ:STAR 0.5E+9; STOP 1E+9
```

Reset the analyzer (activating a frequency sweep) and set the sweep range between 1 GHz and 1.1 GHz.

```
FREQ:CONV:HARM:ORD 4; SPOR 1; RPOR 2
```

Configure a 4th harmonic measurement with source port 1 and receive port 2. The measurement is still not enabled.

```
FREQ:CONV HARM
```

Enable the configured harmonic measurement.

```
CALC:PAR:MEAS? "TRC1"
```

Query the measured quantity. The analyzer measures the wave quantity b2, the response is `'B2D1SAM'`.

```
FREQ:CONV:HARM:RELative ON
```

Enable the relative harmonic measurement, creating a new channel and a new trace *Trc2*.

```
CALC2:PAR:MEAS? "TRC2"
```

Query the measured quantity in the new channel no. 2. The analyzer still measures the wave quantity b2, the response is `'B2D1SAM'`.

```
CALC2:MATH:STAT?
```

Check whether the active trace in channel no. 2 (i.e. *Trc2)* is a mathematical trace: the response is `1`.

## [SENSe<Ch>:]FREQuency:CONVersion:ARBitrary <numerator>, <denominator>, <offset>, CW | FIXed | SWEep

Defines the receiver frequency 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.

NOTE *The frequency formula is applied even if the analyzer returns an error message, because the frequency is outside the allowed range.*

**<Ch>** Channel number.

**<numerator>**
**<denominator>**
**<offset>** Parameters of the frequency formula. The receiver frequency $f_r$ is calculated according to
$f_r$ = <numerator>/<denominator>*$f_b$. + <offset>
where $f_b$ represents the sweep range (for frequency sweeps, parameter SWEep). For power, time and *CW Mode* sweeps (parameters CW or FIXed), $f_b$ = 0.

NOTE *The conversion formula for $f_r$ is ZVR-compatible but differs from manual control where the ratio <numerator>/<denominator> also multiplies the <offset>.*

Range [def. unit] The converted frequency or frequency range must be within the analyzer's frequency range, depending on the instrument model [Hz]. In addition the following restrictions apply:
<numerator> – Values are rounded to positive or negative integer numbers; zero is not allowed [1]
<denominator> – Values are rounded to positive integer numbers [1]
<offset> – Values are rounded to (positive or negative) multiples of 1 Hz [Hz].

*RST value The default frequency/frequency range corresponds to the sweep range or CW frequency of the analyzer, i.e.
<numerator> = <denominator> = 1
<offset> = 0 Hz

**Sweep type**
**Parameters** SWEep – A frequency sweep is active, the command defines a frequency range.
CW | FIXed – A power, time or *CW Mode* sweeps is active, the command defines a fixed frequency.

*RST value SWEep

**SCPI, Command Types** Device-specific, command or query.

**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.

```
SENS:FREQ:CONV:ARB  2, 1, 1E+9, SWE
```

Convert the receiver frequency to the range between 3 GHz and 3.2 GHz.

## [SENSe<Ch>:]FREQuency:CONVersion:ARBitrary:PMETer<Mtr> <numerator>, <denominator>, <offset>, CW | FIXed | SWEep

Defines the receiver frequency 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.

*The frequency formula is applied even if the analyzer returns an error message, because the frequency is outside the allowed range.*

**<Ch>** Channel number.

**<Mtr>** Power meter number

**<numerator>**
**<denominator>**
**<offset>**

Parameters of the frequency formula. The receiver frequency $f_r$ is calculated according to
$f_r$ = <numerator>/<denominator>*$f_b$. + <offset>
where $f_b$ represents the sweep range (for frequency sweeps, parameter SWEep). For power, time and *CW Mode* sweeps (parameters CW or FIXed), $f_b$ = 0.

*The conversion formula for $f_r$ is ZVR-compatible but differs from manual control where the ratio <numerator>/<denominator> also multiplies the <offset>.*

Range [def. unit]

The converted frequency or frequency range must be within the power meter's receive frequency range [Hz]. In addition the following restrictions apply:
<numerator> – Values are rounded to positive or negative integer numbers; zero is not allowed [1]
<denominator> – Values are rounded to positive integer numbers [1]
<offset> – Values are rounded to (positive or negative) multiples of 1 Hz [Hz].

*RST value

The default frequency/frequency range corresponds to the sweep range or CW frequency of the analyzer, i.e.
<numerator> = <denominator> = 1
<offset> = 0 Hz

**Sweep type**
**Parameters**

SWEep – A frequency sweep is active, the command defines a frequency range.
CW | FIXed – A power, time or *CW Mode* sweeps is active, the command defines a fixed frequency.

*RST value

SWEep

**SCPI, Command Types**

Device-specific, command or query.

**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.

## [SENSe<Ch>:]FREQuency:CONVersion:AWReceiver[:STATe] <Boolean>

Qualifies whether a-waves (outgoing waves) are measured at the source or at the receiver frequency. The setting is relevant for all frequency-converting measurements.

| | |
|---|---|
| **<Ch>** | Channel number. |
| **<Boolean>** | OFF - Measurement at source frequency<br>ON - Measurement at receiver frequency |
| *RST value | OFF |
| **SCPI, Command Types** | Device-specific, command or query. |
| **Example:** | `FREQ:CONV:AWR ON` |
| | Measure a waves at the receiver frequency. |

## [SENSe<Ch>:]FREQuency:CONVersion:DEVice:MODE RILI | RILE

Selects the test setup (internal or external sources) for the frequency converter measurement (with option ZVA-K5, Enhanced Frequency Conversion).

| | |
|---|---|
| **<Ch>** | Channel number. This suffix is ignored because the device mode is a global setting. |
| **Parameters** | RILI - Measurement at source frequency<br>RILE - Measurement at receiver frequency |
| *RST value | RILI |
| **SCPI, Command Types** | Device-specific, command or query. |
| **Example:** | `FREQ:CONV:DEV:NAME 'ZVA-Z110'` |
| | Select frequency converter type R&S ZVA-Z110. |
| | `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. |
| | `FREQ:CONV:DEV:MODE RILE` |
| | Measure a waves at the receiver frequency. |

## [SENSe<Ch>:]FREQuency:CONVersion:DEVice:NAME '<Converter Type>'

Selects the frequency converter type for enhanced frequency-converting measurements (with option

ZVA-K5, Enhanced Frequency Conversion).

| | |
|---|---|
| **<Ch>** | Channel number. This suffix is ignored because the converter type is a global setting. |
| **'<Converter Type>'** | 'ZZA-110' - Frequency converter R&S ZZA-110<br>'' - No frequency converter selected |
| *RST value | OFF |
| **SCPI, Command Types** | Device-specific, command or query. |
| **Example:** | `See [SENSe<Ch>:]FREQuency:CONVersion:DEVice:MODE` |

## [SENSe<Ch>:]FREQuency:CONVersion:HARMonic:ORDer <order>

Selects the order of the harmonic measured if a harmonic measurement is enabled (`[SENSe<Ch>:]FREQuency:CONVersion HARMonic`).

| | |
|---|---|
| **<Ch>** | Channel number. |
| **<order>** | Order of the harmonic, integer multiple of the fundamental frequency measured. |
| Range [def. unit] | 2 to 100 [–]. |
| *RST value | 2 |
| **SCPI, Command Types** | Device-specific, command or query. |
| **Example:** | `See [SENSe<Ch>:]FREQuency:CONVersion` |

## [SENSe<Ch>:]FREQuency:CONVersion:HARMonic:RELative <Boolean>

Enables or disables the relative harmonic measurement where the harmonic is divided by the fundamental wave.

NOTE

*With enabled relative measurement the analyzer creates a new channel and a new (mathematical) trace; see program example for `[SENSe<Ch>:]FREQuency:CONVersion`.*

| | |
|---|---|
| **<Ch>** | Channel number. |
| **<Boolean>** | Enable (ON) or disable (OFF) the relative harmonic measurement. |
| *RST value | OFF |
| **SCPI, Command Types** | Device-specific, command or query. |
| **Example:** | `See [SENSe<Ch>:]FREQuency:CONVersion` |

## [SENSe<Ch>:]FREQuency:CONVersion:HARMonic:RPORt <port_no>

Selects the receive port for the harmonic measurement.

| | |
|---|---|
| **<Ch>** | Channel number. |
| **<port_no>** | Integer port number. |

| | |
|---|---|
| Range [def. unit] | Depending on the analyzer model [–]. |
| *RST value | 2 |
| **SCPI, Command Types** | Device-specific, command or query. |
| **Example:** | `See [SENSe<Ch>:]FREQuency:CONVersion` |

## [SENSe<Ch>:]FREQuency:CONVersion:HARMonic:SPORt <port_no> | <ext. gen. no.>

Selects the source port for the harmonic measurement.

| | |
|---|---|
| **<Ch>** | Channel number. |
| **<port_no>** | Positive integer analyzer port number. |
| **<ext. gen. no.>** | Negative integer external generator number (the minus sign serves as an identifier for external generators; –1 denotes the external generator no. 1) |
| Range [def. unit] | Depending on the analyzer model and the number of external generators [–]. |
| *RST value | 1 |
| **SCPI, Command Types** | Device-specific, command or query. |
| **Example:** | `See [SENSe<Ch>:]FREQuency:CONVersion` |

## [SENSe<Ch>:]FREQuency:CONVersion:MIXer...

This subsystem controls the analyzer's mixer measurement mode. Together with the *Harmonic Distortion* measurement and the arbitrary configuration of generator and receiver frequencies *(Port Configuration),* the mixer mode is included in option R&S ZVA-K4.

## [SENSe<Ch>:]FREQuency:CONVersion:MIXer:FFIXed <fixed_frequency>

Defines a fixed frequency, to be assigned either to the RF, the LO, or the IF signal (`[SENSe<Ch>:]FREQuency:CONVersion:MIXer:FIXed RF | LO | IF`).

| | |
|---|---|
| **<Ch>** | Channel number. If unspecified the numeric suffix is set to 1. |
| **<fixed_frequency>** | Fixed frequency. |
| Range [def. unit] | Depending on the instrument model [Hz]. The increment (parameters UP or DOWN) is 0.1 kHz. |
| *RST value | Minimum of the analyzer's frequency range, $f_{MIN}$ |
| **SCPI, Command Types** | Device-specific, command or query. |
| **Example:** | `*RST; FREQ:CONV:MIX:FFIX 1 GHz` |
| | Reset the analyzer and specify a fixed frequency of 1 GHz. |
| | `FREQ:CONV:MIX:FIX RF` |
| | Assign the fixed frequency to the RF signal. The fundamental frequency is |

automatically assigned to the LO signal.

```
FREQ:CONV:MIX:FUND?
```

Query the assignment of the fundamental frequency to prove the last statement. The response is `LO`.

## [SENSe<Ch>:]FREQuency:CONVersion:MIXer:FIXed RF | LO | IF

Selects the mixer input or output signal which is at the fixed frequency defined via (`[SENSe<Ch>:]FREQuency:CONVersion:MIXer:FFIXed`).

| | |
|---|---|
| **<Ch>** | Channel number. |
| **Parameters** | RF – mixer input signal<br>LO – local oscillator signal<br>IF – mixer output signal (mixing product) |
| *RST value | LO |
| **SCPI, Command Types** | Device-specific, command or query. |
| **Example:** | `See SENSe<Ch>:]FREQuency:CONVersion:MIXer:FFIXed` |

## [SENSe<Ch>:]FREQuency:CONVersion:MIXer:FUNDamental RF | LO | IF

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.).

| | |
|---|---|
| **<Ch>** | Channel number. |
| **Parameters** | RF – mixer input signal<br>LO – local oscillator signal<br>IF – mixer output signal (mixing product) |
| *RST value | RF |
| **SCPI, Command Types** | Device-specific, command or query. |
| **Example:** | `See SENSe<Ch>:]FREQuency:CONVersion:MIXer:FFIXed` |

## [SENSe<Ch>:]FREQuency:CONVersion:MIXer:IFFixed <fixed_frequency>

Defines a fixed frequency and assigns it to the IF signal (for alternative commands see example below).

| | |
|---|---|
| **<Ch>** | Channel number. |
| **<fixed_frequency>** | Fixed frequency. |
| Range [def. unit] | Depending on the instrument model [Hz]. The increment (parameters UP or DOWN) is 0.1 kHz. |
| *RST value | Minimum of the analyzer's frequency range, $f_{MIN}$ |
| **SCPI, Command Types** | Device-specific, command or query. |
| **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:LOEXternal NONE | <port_number> | SOURce1 | SOURce2

Selects an external generator as a signal source for the LO signal (external source; to select an internal source use `[SENSe<Ch>:]FREQuency:CONVersion:MIXer:LOINternal`).

| | |
|---|---|
| **<Ch>** | Channel number. |
| **NONE** | LO signal is not controlled by the analyzer. |
| **<port_number>** | Number of external generator as defined in the *System Configuration – External Generators* dialog. |
| *RST value | NONE |
| **SCPI, Command Types** | Device-specific, command or query. |
| **Example:** | `FREQ:CONV:MIX:LOEX 1` |
| | Select the generator no. 1 as a signal source for the LO signal. An error message -222,"Data out of range..." is generated unless an external generator is available. |

## [SENSe<Ch>:]FREQuency:CONVersion:MIXer:LOFixed <fixed_frequency>

Defines a fixed frequency and assigns it to the LO signal (for alternative commands see example below).

| | |
|---|---|
| **<Ch>** | Channel number. |
| **<fixed_frequency>** | Fixed frequency. |
| Range [def. unit] | Depending on the instrument model [Hz]. The increment (parameters UP or DOWN) is 0.1 kHz. |
| *RST value | Minimum of the analyzer's frequency range, $f_{MIN}$ |
| **SCPI, Command Types** | Device-specific, command or query. |
| **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:LOINternal NONE | <port_number>

Selects an analyzer port as a signal source for the LO signal (internal source; to select an external generator use `[SENSe<Ch>:]FREQuency:CONVersion:MIXer:LOEXternal`).

| | |
|---|---|
| **<Ch>** | Channel number. |
| **NONE** | LO signal is not controlled by the analyzer. |
| **<port_number>** | Analyzer port number |
| Range [def. unit] | 3 (for 3-port analyzers), 3, 4 (for 4-port analyzers)<br>No internal source is available for 2-port instruments. |
| *RST value | NONE |
| **SCPI, Command Types** | Device-specific, command or query. |
| **Example:** | `FREQ:CONV:MIX:LOIN 3` |
| | Select port no. 3 as a signal source for the LO signal. |

## [SENSe<Ch>:]FREQuency:CONVersion:MIXer:RFFixed <fixed_frequency>

Defines a fixed frequency and assigns it to the RF signal (for alternative commands see example below).

| | |
|---|---|
| **<Ch>** | Channel number. |
| **<fixed_frequency>** | Fixed frequency. |
| Range [def. unit] | Depending on the instrument model [Hz]. The increment (parameters UP or DOWN) is 0.1 kHz. |
| *RST value | Minimum of the analyzer's frequency range, $f_{MIN}$ |
| **SCPI, Command Types** | Device-specific, command or query. |
| **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:CONVersion:MIXer:TFRequency DCLower | DCUPper | UCONversion | BAND1 | BAND2

Selects the frequency conversion mode of the IF signal.

| | |
|---|---|
| **<Ch>** | Channel number. |
| DCLower | IF = LO – RF (down conversion lower sideband, equivalent to BAND1) |
| DCUPper | IF = RF – LO (down conversion upper sideband, equivalent to BAND2) |
| UCONVersion | IF = LO + RF (up conversion) |
| *RST value | DCUPper |

**SCPI, Command Types** Device-specific, command or query.

**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.

## [SENSe<Ch>:]FREQuency...

This subsystem sets frequency-related parameters, especially the measurement and display ranges for the different sweep types, and controls the harmonic distortion measurement.

***Note:*** *For a frequency sweep the range can be defined alternatively by a combination of Start/Stop frequencies or a Center frequency and Span. In fixed frequency mode (power sweep or Time/CW Mode; see [SENSe<Chn>:]FUNCtion[:ON]) the measurement frequency is set by means of [SENSe<Ch>:]FREQuency:CW|FIXed.*

## [SENSe<Ch>:]FREQuency:CW|FIXed <CW_frequency>

Defines the fixed (Continuous Wave, CW) frequency for all sweep types operating at fixed frequency (power sweep, time sweep, CW mode sweep). The two command forms `FREQ:CW` and `FREQ:FIX` are equivalent.

`[SENSe<Ch>:]FREQuency:CW|FIXed` is equivalent to `SOURce<Ch>:FREQuency<Pt>:CW|FIXed`. Source and receiver frequency are equal; the four commands overwrite each other. With option ZVA-K4, *Frequency Conversion,* independent offset frequencies can be defined for the source and receiver frequencies; see `SOURce<Ch>:FREQuency<Pt>:CONVersion:ARBitrary:IFRequency` and `[SENSe<Ch>:]FREQuency:CONVersion:ARBitrary`.

**<Ch>** Channel number.

**<CW_frequency>** Fixed stimulus and analyzer frequency.

Range [def. unit] Depending on the instrument model [Hz]. The increment (parameters UP or DOWN) is 0.1 kHz.

*RST value 1 GHz

**SCPI, Command Types** Confirmed, command or query.

**Example:**

```
FUNC "XPOW:POW:A1"
```

Activate a power sweep and select the wave quantity a1 `as measured parameter for channel and trace no. 1`.

```
FREQ:CW 100MHz
```

Set the CW frequency to 100 MHz.

## [SENSe<Ch>:]FREQuency:LPNoise <Boolean>

Enables or disables the *Low Phase Noise* function.

| | |
|---|---|
| **<Ch>** | Channel number. |
| **<Boolean>** | ON \| OFF - Low phase noise on or off. |
| *RST value | OFF |
| **SCPI, Command Types** | Device-specific, command or query. |
| **Example:** | `FREQ:LPN ON` |
| | Enable the *Low Phase Noise* function. |

## [SENSe<Ch>:]FREQuency:MODE SWEep | SEGMent | CW | FIXed

Selects the sweep type and defines which set of commands controls the stimulus frequency.

The command `[SENSe<Ch>:]SWEep:TYPE` provides a complete list of sweep types.

| | |
|---|---|
| **<Ch>** | Channel number. |
| **SWEep** | 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. |
| **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. |
| **CW \| FIXed** | Time sweep. The measurement is performed at constant frequency (`[SENSe<Ch>:]FREQuency:CW\|FIXed`) and repeated over a specified period of time at constant time intervals. |
| *RST value | SWEep |
| **SCPI, Command Types** | Confirmed (with device-specific *RST value and parameters), command or query. |
| **Example:** | `FREQ:MODE CW` |
| | Activate a time sweep. |
| | `FREQ:CW 100MHz` |
| | Set the CW frequency to 100 MHz. |

## [SENSe<Ch>:]FREQuency:SBANd POSitive | NEGative | AUTO

Defines whether the analyzer measures with a local oscillator frequency LO below or above the RF input frequency.

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.

| | |
|---|---|
| **<Ch>** | Channel number. |
| **POSitive** | The LO frequency is always above the measured RF frequency. |
| **NEGative** | The LO frequency is always below the measured RF frequency. |
| **AUTO** | The analyzer auto-selects the local oscillator frequency, depending on the receiver (RF) frequency and the test port. |
| *RST value | AUTO |
| **SCPI, Command Types** | Device-specific, command or query. |
| **Example:** | `*RST; FREQ:TYPE?; :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). |

## [SENSe<Ch>:]FREQuency:SPAN 

Defines the width of the measurement and display range for a frequency sweep.

| | |
|---|---|
| **<Ch>** | Channel number. |
| **** | Frequency span of the sweep. |
| Range [def. unit] | Depending on the instrument model [Hz]. The increment (parameters UP or DOWN) is 0.1 kHz. |
| *RST value | Maximum frequency range of the analyzer: $f_{MAX}$ - $f_{MIN}$ |
| **SCPI, Command Types** | Confirmed, command or query. |
| **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. |
| | `FREQ:CENT 100MHz` |
| | Set center frequency to 100 MHz. |
| | `FREQ:SPAN 50000` |
| | Set frequency span to 50 kHz, leaving the center frequency unchanged. |

***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 <start_frequency>

Defines the start frequency for a frequency sweep which is equal to the left edge of a Cartesian diagram.

| | |
|---|---|
| **<Ch>** | Channel number. |
| **<start_frequency>** | Start frequency of the sweep. |
| Range [def. unit] | Depending on the instrument model [Hz]. The increment (parameters UP or DOWN) is 0.1 kHz. |
| *RST value | Minimum frequency of the analyzer: MIN |
| **SCPI, Command Types** | Confirmed, command or query. |
| **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. |
| | `FREQ:STAR 100000` |
| | Set start frequency to 100 kHz. |
| | `FREQ:STOP 10MHz` |
| | Set stop frequency to 10 MHz. |

***Note:*** *If the start frequency entered is greater than the current stop frequency ([SENSe<Ch>:]FREQuency:STOP), the stop frequency is set to the start frequency plus the minimum frequency span ([SENSe<Ch>:]FREQuency:SPAN).*

## [SENSe<Ch>:]FREQuency:STOP <stop_frequency>

Defines the stop frequency for a frequency sweep which is equal to the right edge of a Cartesian diagram.

| | |
|---|---|
| **<Ch>** | Channel number. |
| **<stop_frequency>** | Stop frequency of the sweep. |
| Range [def. unit] | Depending on the instrument model [Hz]. The increment (parameters UP or DOWN) is 0.1 kHz. |
| *RST value | Maximum frequency of the analyzer: MAX |
| **SCPI, Command Types** | Confirmed, command or query. |
| **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. |
| | `FREQ:STAR 100000` |
| | Set start frequency to 100 kHz. |
| | `FREQ:STOP 10MHz` |
| | Set stop frequency to 10 MHz. |

***Note:*** *If the stop frequency entered is smaller than the current start frequency ([SENSe<Ch>:]FREQuency:STARt), the start frequency is set to the stop frequency minus the minimum frequency span ([SENSe<Ch>:]FREQuency:SPAN).*

## [SENSe<Chn>:]FUNCtion...

This subsystem selects the sweep type and the measurement parameter.

## [SENSe<Chn>:]FUNCtion[:ON] '<string>'

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.*

| | |
|---|---|
| **<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. |
| **'<string>'** | Single string parameter defining the sweep type and the parameter to be measured: <string> = "<sweep_type>:<parameter>". |
| Range [def. unit] | See list of strings below [–]. |
| *RST value | "XFR:S21" ' |
| **SCPI, Command Types** | Confirmed, command or query |

**Example:**

```
CALC4:PAR:SDEF "Ch4Tr1", "S11"
```

Create channel 4 and a trace named *Ch4Tr1* to measure the input reflection coefficient S11. 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<Pt`$_{out}$`><Pt`$_{in}$`>` | S-parameter with output and input port number of the DUT, e.g. `S11, S21`. |
| `POWer:RATio A<Pt`$_{out}$`> \| B<Pt`$_{in}$`>, A<Pt`$_{out}$`> \| B<Pt`$_{in}$`>`<br>Output: `A<Pt`$_{out}$`>/B<Pt`$_{in}$`>` | Ratio, e.g. `B2, A1` for b2/a1 drive Port 1 |
| `POWer:A<Pt`$_{out}$`>` | Wave quantity with stimulus port number of the analyzer, e.g. `A1`. |
| `POWer:B<Pt`$_{in}$`>` | Wave quantity with receive port number of the analyzer, e.g. `B2`. |
| `POWer:Z<Pt`$_{out}$`><Pt`$_{in}$`>` | Matched-circuit impedance (converted Z-parameter) with output and input port number of the DUT, e.g. `Z11, Z21`. |
| `POWer:Y<Pt`$_{out}$`><Pt`$_{in}$`>` | Matched-circuit admittance (converted Y-parameter) with output and input port number of the DUT, e.g. `Y11, Y21`. |
| `POWer:KFACtor`<br>`POWer:MUFactor<1\|2>` | Stability factor K<br>Stability factors μ1 or μ2 |

| | |
|---|---|
| PAE<Pt$_{out}$><Pt$_{in}$> | Power Added Efficiency (PAE) with output and input port number of the DUT, e.g. PAE21 (output variable for query only) |
| VOLTage[:DC] DCIN1 \| DCIN2<br>Output: DC ±1 V, DC ±10 V | DC Input 1 (DC ±1 V) or DC Input 2 (DC ±10 V) |

***Note:** The mnemonics POWer: and VOLTage: are not used in output strings.*

## [SENSe<Ch>:]LO...

This subsystem controls the Local Oscillator (LO) amplifiers for the reference and measurement channels.

## [SENSe<Ch>:]LOMeasure<Pt>[:STATe] <Boolean>

Switches the Local Oscillator amplifier for the measurement channel b on or off.

**<Ch>** Channel number. If unspecified the numeric suffix is set to 1.

**<Pt>** Test port number of the analyzer.

**<Boolean>** ON | OFF – Switches the LO oscillator on or off.
*RST value ON

**SCPI, Command Types** Device-specific, command or query

**Example:**

```
LOM1 OFF; LOR1 OFF
```

Switch the local oscillators for the measurement channel b and the reference channel a at port no. 1 off.

## [SENSe<Ch>:]LOReference[:STATe] <Boolean>

Switches the Local Oscillator amplifier for the reference channel a on or off.

**<Ch>** Channel number.

**<Pt>** Test port number of the analyzer.

**<Boolean>** ON | OFF – Switches the LO oscillator on or off.
*RST value ON

**SCPI, Command Types** Device-specific, command or query

**Example:**

```
LOM1 OFF; LOR1 OFF
```

Switch the local oscillators for the measurement channel b and the reference channel a at port no. 1 off.

## [SENSe<Ch>:]LPORt...

This subsystem defines the reference impedances of the balanced ports.

## [SENSe<Ch>:]LPORt<log_port>:ZCOMmon <real>[, <imaginary>]

Specifies the complex common mode reference impedance for the balanced (logical) port numbered <log_port>.

**<Ch>** Channel number. If unspecified the numeric suffix is set to 1.

**<log_port>** Logical port number. The logical ports must be defined using `SOURce<Ch>:LPORt<log_port> <phys_port1>,<phys_port2>`. An n port analyzer supports a maximum of n/2 (n even) or (n – 1)/2 (n odd) logical ports.

**<real>**
**<imaginary>** Real and imaginary part of the port impedance. <imaginary> may be omitted to define a real impedance.

Range [def. unit] 1mΩ to 10 MΩ for the real part [Ω]
– 10 MΩ to 10 MΩ for the imaginary part [Ω]

*RST value 25 Ω (real impedance)

**SCPI, Command Types** Device-specific, command or query

**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:ZCOM 27, 2
```

Specify a complex common mode reference impedance of 27Ω + j * 2Ω for the defined port.

## [SENSe<Ch>:]LPORt<log_port>:ZDIFferent <real>[, <imaginary>]

Specifies the complex differential mode reference impedance for the balanced (logical) port numbered <log_port>.

**<Ch>** Channel number. If unspecified the numeric suffix is set to 1.

**<log_port>** Logical port number. The logical ports must be defined using `SOURce<Ch>:LPORt<log_port> <phys_port1>,<phys_port2>`. An n port analyzer supports a maximum of n/2 (n even) or (n – 1)/2 (n odd) logical ports.

**<real>**
**<imaginary>** Real and imaginary part of the port impedance. <imaginary> may be omitted to define a real impedance.

Range [def. unit] 1mΩ to 10 MΩ for the real part [Ω]
– 10 MΩ to 10 MΩ for the imaginary part [Ω]

| | |
|---|---|
| *RST value | 100 Ω (real impedance) |
| **SCPI, Command Types** | Device-specific, command or query |
| **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. |

## [SENSe<Chn>:]PAE...

This subsystem configures the measurement of the Power Added Efficiency (PAE).

## [SENSe<Chn>:]PAE:C <numeric_value>

Defines the constant c for the PAE measurement.

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace. |
| **<numeric_value>** | Value of the constant c in the PAE test model (`[SENSe<Chn>:]PAE:EXPRession`). The value is used if one of the test models `C10 \| C1 \| CK11` is selected. |
| Range [def. unit] | -3.4028234664E+038 W/V to +3.4028234664E+038 W/V [W/V]. |
| *RST value | 0 W/V |
| **SCPI, Command Types** | Device-specific, command or query |
| **Example:** | `*RST; CALC:PAR:MEAS "TRC1", "PAE21"` |
| | Select the PAE as a measurement parameter for the default trace *Trc1*. Select port 1 as the DUT input port, port 2 as the DUT output port. |
| | `PAE:EXPR CK11` |
| | Select the test model involving both constants and no power approximation. |
| | `PAE:C 10; K 1` |
| | Set the constant c = $U_{DC}$ / R to 10 W/V and the constant k = 1 W/V$^2$. If the test setup is in accordance with the settings made, the trace shows the PAE for a constant DC supply voltage $U_{DC}$. |

## [SENSe<Chn>:]PAE:EXPRession C10 | C1 | K101 | CK11

Selects the test model for the PAE measurement.

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace. |
| **C10** | The DC power is equal to the constant c (`[SENSe<Chn>:]PAE:C`) times the DC voltage measured at the DC input *DC MEAS ± 10 V*. |
| **C1** | The DC power is equal to the constant c times the DC voltage measured at the DC input *DC MEAS ± 1 V*. |
| **K101** | The DC power is equal to the constant k (`[SENSe<Chn>:]PAE:K`) times the DC voltage measured at the DC input *DC MEAS ± 10 V* times the DC voltage measured at the DC input *DC MEAS ± 1 V*. |
| **CK11** | The DC power is equal to the constant c times the DC voltage measured at the DC input *DC MEAS ± 1 V* minus k times the square of the DC voltage measured at the DC input *DC MEAS ± 1 V*. |
| *RST value | CDC10 |
| **SCPI, Command Types** | Device-specific, command or query |
| **Example:** | `See [SENSe<Chn>:]PAE:C` |

## [SENSe<Chn>:]PAE:K <numeric_value>

Defines the constant k for the PAE measurement.

| | |
|---|---|
| **<Chn>** | Channel number used to identify the active trace. |
| **<numeric_value>** | Value of the constant k in the PAE test model (`[SENSe<Chn>:]PAE:EXPRession`). The value is used if one of the test models `K101 \| CK11` is selected. |
| Range [def. unit] | -3.4028234664E+038 $W/V^2$ to +3.4028234664E+038 $W/V^2$ [$W/V^2$]. |
| *RST value | 0 W/V |
| **SCPI, Command Types** | Device-specific, command or query |
| **Example:** | `See [SENSe<Chn>:]PAE:C` |

## [SENSe<Ch>:]PORt...

This subsystem controls the frequency reference oscillator.

## [SENSe<Ch>:]PORt<phys_port>:ZREFerence <real>[, <imaginary>]

Specifies the complex reference impedance for the physical port numbered <phys_port>.

| | |
|---|---|
| **<Ch>** | Channel number. If unspecified the numeric suffix is set to 1. |
| **<phys_port>** | Physical port number. The number of ports depends on the analyzer model. |
| **<real>**<br>**<imaginary>** | Real and imaginary part of the port impedance. <imaginary> may be omitted to define a real impedance. |
| Range [def. unit] | 1mΩ to 10 MΩ for the real part [Ω]<br>– 10 MΩ to 10 MΩ for the imaginary part [Ω] |

| | |
|---|---|
| *RST value | Default reference impedance of the connector family assigned to the port (real impedance, e.g. 50 Ω). |
| **SCPI, Command Types** | Device-specific, command or query |
| **Example:** | `POR2:ZREF 52, 2` |
| | Specify a complex reference impedance of 52 Ω + j * 2 Ω for the (physical) port no. 2. |

## [SENSe<Ch>:]POWer...

This subsystem controls the power at the input ports of the analyzer.

## [SENSe<Ch>:]POWer:ATTenuation ARECeiver | BRECeiver | CRECeiver | DRECeiver, <numeric_value>

Sets an attenuation factor for the wave received at test port no. <Pt>. The generated wave is attenuated via SOURce:POWer<Port_no>:ATTenuation.

| | |
|---|---|
| **<Ch>** | Channel number. If unspecified the numeric suffix is set to 1. |
| **ARECeiver \| BRECeiver \| CRECeiver \| DRECeiver** | Test port of the analyzer, depending on the analyzer model:<br>Test port 1: ARECeiver<br>Test port 2: BRECeiver<br>Test port 3: CRECeiver<br>Test port 4: DRECeiver |
| **<numeric_value>** | Attenuation factor for the received wave. |
| Range [def. unit] | Depending on the analyzer/attenuator model, e.g. 0 dB to 75 dB for ZVA8/ZVA8-B31/32/B33/B34 [dB]. UP and DOWN increment/decrement the attenuation in 5 or 10 dB steps. The analyzer rounds any entered value below the maximum attenuation to the closest step. |
| *RST value | 0 dB |
| **SCPI, Command Types** | Confirmed, command or query |
| **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. |

## [SENSe<Ch>:]POWer:IFGain<Pt>:MEASure AUTO | LNOise | LDIStortion

Selects the IF gain in the measurement channel b.

| | |
|---|---|
| **<Ch>** | Channel number. |
| **<Pt>** | Test port number of the analyzer. |
| **Parameters** | AUTO – Adaptive Gain Control according to the RF input level<br>LNOise – Fixed, small IF gain, for high input levels |

LDIStortion – Fixed, large IF gain, for low input levels

*RST value AUTO

**SCPI, Command Types** Device-specific, command or query

**Example:**

```
POWer:IFG2:MEAS LDIS; POWer:IFG2:REF LNO
```

Optimize the measurement channel b for small input levels, the reference channel a for high input levels.

## [SENSe<Ch>:]PULSe...

This subsystem controls pulsed measurements (with option ZVA-K7, Measurements on Pulsed Signals).

## [SENSe<Ch>:]PULSe:COUPled[:STATe] <Boolean>

Couples the section limits for averaging (and the section limit lines) for all receivers and source ports and in all channels.

**<Ch>** Channel number. If unassigned this value is set to 1.

**<Boolean>** ON | OFF - Section limits coupled or uncoupled
*RST value OFF

**SCPI, Command Types** Device-specific, command or query

**Example:**

```
*RST; SWE:TYPE PULS; :PULS:COUP ON
```

Select pulse profile mode and activate coupled section limits.

```
SENS2:PULS:REC:A1:SRCP1:EVAL:STAR 200 ns
```

Create channel 2 and define a start value of 200 ns for the averaging section for the $a_1$ receiver and source port no. 1.

```
SENS1:PULS:REC:A1:SRCP1:EVAL:STAR?
```

Query the section start value for channel 1. As the sections are coupled, the response is 2E-007.

## [SENSe<Ch>:]PULSe:RECeiver:A<rec_no>:GENerator<gen_no>:EVALuation:MODE NORMal | MEAN

Specifies whether the wave quantity assigned to the reference receiver <rec_no> and the external generator <gen_no> is displayed as measured or whether it is averaged.

**<Ch>** Channel number

**<rec_no>** Number of the reference receiver (= port number of the analyzer). Values range from 1 to the total number of test ports.

**<gen_no>** Number of a previously configured external generator

| | |
|---|---|
| **Parameters** | NORMal – wave quantity is as measured (pulse shape)<br>MEAN – wave quantity is replaced by its mean value (magnitude and phase) averaged over a configurable time interval; see example below |
| *RST value | NORMal |
| **SCPI, Command Types** | Device-specific, command or query |
| **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.

```
PULS:REC:A2:GEN1:EVAL:STAR 200 ns; STOP 500 ns
```

Specify the averaging section for the $a_2$ receiver and the configured generator: Define a start value of 200 ns and a stop value of 500 ns.

```
PULS:REC:A2:GEN1:LIN ON
```

Display the section limit lines in the diagram.

```
PULS:REC:A2:GEN1:EVAL:MODE MEAN
```

Enable averaging of the $a_2$ wave.

```
PULS:REC:A2:GEN1:TRIG:DEL 100 ns
```

Shift the stimulus axis of the $a_2$ wave by 100 ns.

## [SENSe<Ch>:]PULSe:RECeiver:A<rec_no>:GENerator<gen_no>:EVALuation:STARt <section_start>

Defines the start time of the averaging section for the wave quantity assigned to the reference receiver <rec_no> and the external generator <gen_no>.

| | |
|---|---|
| **<Ch>** | Channel number |
| **<rec_no>** | Number of the reference receiver (= port number of the analyzer). Values range from 1 to the total number of test ports. |
| **<gen_no>** | Number of a previously configured external generator |
| **<section_start>** | Start time of the section |
| Range [def. unit] | –0.003 s to +0.002999975 s [s]. The increment (UP/DOWN) is 50 ns. The width of the averaging section must be between 25 ns and 3 s. |
| *RST value | 0 s |
| **SCPI, Command Types** | Device-specific, command or query |
| **Example:** | `See [SENSe<Ch>:]PULSe:RECeiver:A<rec_no>:GENerator<gen_no>:EVALuation:MODE` |

## [SENSe<Ch>:]PULSe:RECeiver:A<rec_no>:GENerator<gen_no>:EVALuation:STOP <section_stop>

Defines the stop time of the averaging section for the wave quantity assigned to the reference receiver <rec_no> and the external generator <gen_no>.

| | |
|---|---|
| **<Ch>** | Channel number |
| **<rec_no>** | Number of the reference receiver (= port number of the analyzer). Values range from 1 to the total number of test ports. |
| **<gen_no>** | Number of a previously configured external generator |
| **<section_stop>**<br>Range [def. unit] | Stop time of the section<br>–0.002999975 s to +0.003 s [s]. The increment (UP/DOWN) is 50 ns. The width of the averaging section must be between 25 ns and 3 s. |
| *RST value | 0 s |
| **SCPI, Command Types** | Device-specific, command or query |
| **Example:** | `See [SENSe<Ch>:]PULSe:RECeiver:A<rec_no>:GENerator<gen_no>:EVALuation:MODE` |

## [SENSe<Ch>:]PULSe:RECeiver:A<rec_no>:GENerator<gen_no>:LINes[:STATe] <Boolean>

Displays or hides the limit lines of the averaging section for the wave quantity assigned to the reference receiver <rec_no> and the external generator <gen_no>.

| | |
|---|---|
| **<Ch>** | Channel number |
| **<rec_no>** | Number of the reference receiver (= port number of the analyzer). Values range from 1 to the total number of test ports. |
| **<gen_no>** | Number of a previously configured external generator |
| **<Boolean>**<br>*RST value | ON \| OFF – Display or hide lines<br>OFF |
| **SCPI, Command Types** | Device-specific, command or query |
| **Example:** | `See [SENSe<Ch>:]PULSe:RECeiver:A<rec_no>:GENerator<gen_no>:EVALuation:MODE` |

## [SENSe<Ch>:]PULSe:RECeiver:A<rec_no>:GENerator<gen_no>:TRIGger:DELay <shift_stimulus>

Defines an offset time (shift stimulus) for the wave quantity assigned to the reference receiver <rec_no> and the external generator <gen_no>.

| | |
|---|---|
| **<Ch>** | Channel number |
| **<rec_no>** | Number of the reference receiver (= port number of the analyzer). Values range from 1 to the total number of test ports. |
| **<gen_no>** | Number of a previously configured external generator |
| **<shift_stimulus>**<br>Range [def. unit]<br>*RST value | Offset time<br>–0.003 s to +0.003 s [s]. The increment (UP/DOWN) is 50 ns.<br>0 s |
| **SCPI, Command** | Device-specific, command or query |

**Types**
**Example:**

```
See
[SENSe<Ch>:]PULSe:RECeiver:A<rec_no>:GENerator<gen_no>:EVALuation:MODE
```

## [SENSe<Ch>:]PULSe:RECeiver:A<rec_no>:SRCPort<port_no>:EVALuation:MODE NORMal | MEAN

Specifies whether the wave quantity assigned to the reference receiver <rec_no> and the analyzer source port <port_no> is displayed as measured or whether it is averaged.

**<Ch>** Channel number

**<rec_no>** Number of the reference receiver (= port number of the analyzer). Values range from 1 to the total number of test ports.

**<port_no>** Number of the analyzer source port. 1 to the total number of test ports.

**Parameters** NORMal – wave quantity is as measured (pulse shape)
MEAN – wave quantity is replaced by its mean value (magnitude and phase) averaged over a configurable time interval; see example below

*RST value NORMal

**SCPI, Command** Device-specific, command or query

**Types**
**Example:**

```
PULS:REC:A2:SRCP2:EVAL:STAR 200 ns; STOP 500 ns
```

Specify the averaging section for the $a_2$ receiver and the analyzer source port no. 2: Define a start value of 200 ns and a stop value of 500 ns.

```
PULS:REC:A2:SRCP2:LIN ON
```

Display the section limit lines in the diagram.

```
PULS:REC:A2:SRCP2:EVAL:MODE MEAN
```

Enable averaging of the $a_2$ wave.

```
PULS:REC:A2:SRCP2:TRIG:DEL 100 ns
```

Shift the stimulus axis of the $a_2$ wave by 100 ns.

## [SENSe<Ch>:]PULSe:RECeiver:A<rec_no>:SRCPort<port_no>:EVALuation:STARt <section_start>

Defines the start time of the averaging section for the wave quantity assigned to the reference receiver <rec_no> and the analyzer source port <port_no>.

**<Ch>** Channel number

**<rec_no>** Number of the reference receiver (= port number of the analyzer). Values range from 1 to the total number of test ports.

**<port_no>** Number of the analyzer source port. 1 to the total number of test ports.

**<section_start>** Start time of the section

Range [def. unit] –0.003 s to +0.002999975 s [s]. The increment (UP/DOWN) is 50 ns. The width of the averaging section must be between 25 ns and 3 s.

*RST value 0 s

| | |
|---|---|
| **SCPI, Command Types** | Device-specific, command or query |
| **Example:** | `See [SENSe<Ch>:]PULSe:RECeiver:A<rec_no>:SRCPort<gen_no>:EVALuation:MODE` |

## [SENSe<Ch>:]PULSe:RECeiver:A<rec_no>:SRCPort<port_no>:EVALuation:STOP <section_stop>

Defines the stop time of the averaging section for the wave quantity assigned to the reference receiver <rec_no> and the analyzer source port <port_no>.

| | |
|---|---|
| **<Ch>** | Channel number |
| **<rec_no>** | Number of the reference receiver (= port number of the analyzer). Values range from 1 to the total number of test ports. |
| **<port_no>** | Number of the analyzer source port. 1 to the total number of test ports. |
| **<section_stop>** | Stop time of the section |
| Range [def. unit] | –0.002999975 s to +0.003 s [s]. The increment (UP/DOWN) is 50 ns. The width of the averaging section must be between 25 ns and 3 s. |
| *RST value | 0 s |
| **SCPI, Command Types** | Device-specific, command or query |
| **Example:** | `See [SENSe<Ch>:]PULSe:RECeiver:A<rec_no>:SRCPort<gen_no>:EVALuation:MODE` |

## [SENSe<Ch>:]PULSe:RECeiver:A<rec_no>:SRCPort<port_no>:LINes[:STATe] <Boolean>

Displays or hides the limit lines of the averaging section for the wave quantity assigned to the reference receiver <rec_no> and the analyzer source port <port_no>.

| | |
|---|---|
| **<Ch>** | Channel number |
| **<rec_no>** | Number of the reference receiver (= port number of the analyzer). Values range from 1 to the total number of test ports. |
| **<port_no>** | Number of the analyzer source port. 1 to the total number of test ports. |
| **<Boolean>** | ON \| OFF – Display or hide lines |
| *RST value | OFF |
| **SCPI, Command Types** | Device-specific, command or query |
| **Example:** | `See [SENSe<Ch>:]PULSe:RECeiver:A<rec_no>:SRCPort<gen_no>:EVALuation:MODE` |

## [SENSe<Ch>:]PULSe:RECeiver:A<rec_no>:SRCPort<port_no>:TRIGger:DELay <shift_stimulus>

Defines an offset time (shift stimulus) for the wave quantity assigned to the reference receiver <rec_no>

and the analyzer source port <port_no>.

| | |
|---|---|
| **<Ch>** | Channel number |
| **<rec_no>** | Number of the reference receiver (= port number of the analyzer). Values range from 1 to the total number of test ports. |
| **<port_no>** | Number of the analyzer source port. 1 to the total number of test ports. |
| **<shift_stimulus>** | Offset time |
| Range [def. unit] | –0.003 s to +0.003 s [s]. The increment (UP/DOWN) is 50 ns. |
| *RST value | 0 s |
| **SCPI, Command Types** | Device-specific, command or query |
| **Example:** | `See [SENSe<Ch>:]PULSe:RECeiver:A<rec_no>:SRCPort<gen_no>:EVALuation:MODE` |

## [SENSe<Ch>:]PULSe:RECeiver:B<rec_no>:GENerator<gen_no>:EVALuation:MODE NORMal | MEAN

Specifies whether the wave quantity assigned to the measurement receiver <rec_no> and the external generator <gen_no> is displayed as measured or whether it is averaged.

| | |
|---|---|
| **<Ch>** | Channel number |
| **<rec_no>** | Number of the measurement receiver (= port number of the analyzer). Values range from 1 to the total number of test ports. |
| **<gen_no>** | Number of a previously configured external generator |
| **Parameters** | NORMal – wave quantity is as measured (pulse shape)<br>MEAN – wave quantity is replaced by its mean value (magnitude and phase) averaged over a configurable time interval; see example below |
| *RST value | NORMal |
| **SCPI, Command Types** | Device-specific, command or query |
| **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.

```
PULS:REC:B2:GEN1:EVAL:STAR 200 ns; STOP 500 ns
```

Specify the averaging section for the $b_2$ receiver and the configured generator: Define a start value of 200 ns and a stop value of 500 ns.

```
PULS:REC:B2:GEN1:LIN ON
```

Display the section limit lines in the diagram.

```
PULS:REC:B2:GEN1:EVAL:MODE MEAN
```

Enable averaging of the $b_2$ wave.

```
PULS:REC:B2:GEN1:TRIG:DEL 100 ns
```

Shift the stimulus axis of the $b_2$ wave by 100 ns.

## [SENSe<Ch>:]PULSe:RECeiver:B<rec_no>:GENerator<gen_no>:EVALuation:ST

### ARt <section_start>

Defines the start time of the averaging section for the wave quantity assigned to the measurement receiver <rec_no> and the external generator <gen_no>.

| | |
|---|---|
| **<Ch>** | Channel number |
| **<rec_no>** | Number of the measurement receiver (= port number of the analyzer). Values range from 1 to the total number of test ports. |
| **<gen_no>** | Number of a previously configured external generator |
| **<section_start>** | Start time of the section |
| Range [def. unit] | –0.003 s to +0.002999975 s [s]. The increment (UP/DOWN) is 50 ns. The width of the averaging section must be between 25 ns and 3 s. |
| *RST value | 0 s |
| **SCPI, Command Types** | Device-specific, command or query |
| **Example:** | `See [SENSe<Ch>:]PULSe:RECeiver:B<rec_no>:GENerator<gen_no>:EVALuation:MODE` |

### [SENSe<Ch>:]PULSe:RECeiver:B<rec_no>:GENerator<gen_no>:EVALuation:STOP <section_stop>

Defines the stop time of the averaging section for the wave quantity assigned to the measurement receiver <rec_no> and the external generator <gen_no>.

| | |
|---|---|
| **<Ch>** | Channel number |
| **<rec_no>** | Number of the measurement receiver (= port number of the analyzer). Values range from 1 to the total number of test ports. |
| **<gen_no>** | Number of a previously configured external generator |
| **<section_stop>** | Stop time of the section |
| Range [def. unit] | –0.002999975 s to +0.003 s [s]. The increment (UP/DOWN) is 50 ns. The width of the averaging section must be between 25 ns and 3 s. |
| *RST value | 0 s |
| **SCPI, Command Types** | Device-specific, command or query |
| **Example:** | `See [SENSe<Ch>:]PULSe:RECeiver:B<rec_no>:GENerator<gen_no>:EVALuation:MODE` |

### [SENSe<Ch>:]PULSe:RECeiver:B<rec_no>:GENerator<gen_no>:LINes[:STATe] <Boolean>

Displays or hides the limit lines of the averaging section for the wave quantity assigned to the measurement receiver <rec_no> and the external generator <gen_no>.

| | |
|---|---|
| **<Ch>** | Channel number |
| **<rec_no>** | Number of the measurement receiver (= port number of the analyzer). Values |

| | |
|---|---|
| | range from 1 to the total number of test ports. |
| **<gen_no>** | Number of a previously configured external generator |
| **<Boolean>** | ON \| OFF – Display or hide lines |
| *RST value | OFF |
| **SCPI, Command Types** | Device-specific, command or query |
| **Example:** | `See [SENSe<Ch>:]PULSe:RECeiver:B<rec_no>:GENerator<gen_no>:EVALuation:MODE` |

## [SENSe<Ch>:]PULSe:RECeiver:B<rec_no>:GENerator<gen_no>:TRIGger:DELay <shift_stimulus>

Defines an offset time (shift stimulus) for the wave quantity assigned to the measurement receiver <rec_no> and the external generator <gen_no>.

| | |
|---|---|
| **<Ch>** | Channel number |
| **<rec_no>** | Number of the measurement receiver (= port number of the analyzer). Values range from 1 to the total number of test ports. |
| **<gen_no>** | Number of a previously configured external generator |
| **<shift_stimulus>** | Offset time |
| Range [def. unit] | –0.003 s to +0.003 s [s]. The increment (UP/DOWN) is 50 ns. |
| *RST value | 0 s |
| **SCPI, Command Types** | Device-specific, command or query |
| **Example:** | `See [SENSe<Ch>:]PULSe:RECeiver:B<rec_no>:GENerator<gen_no>:EVALuation:MODE` |

## [SENSe<Ch>:]PULSe:RECeiver:B<rec_no>:SRCPort<port_no>:EVALuation:MODE NORMal | MEAN

Specifies whether the wave quantity assigned to the measurement receiver <rec_no> and the analyzer source port <port_no> is displayed as measured or whether it is averaged.

| | |
|---|---|
| **<Ch>** | Channel number |
| **<rec_no>** | Number of the measurement receiver (= port number of the analyzer). Values range from 1 to the total number of test ports. |
| **<port_no>** | Number of the analyzer source port. 1 to the total number of test ports. |
| **Parameters** | NORMal – wave quantity is as measured (pulse shape)<br>MEAN – wave quantity is replaced by its mean value (magnitude and phase) averaged over a configurable time interval; see example below |
| *RST value | NORMal |
| **SCPI, Command Types** | Device-specific, command or query |
| **Example:** | `PULS:REC:B2:SRCP2:EVAL:STAR 200 ns; STOP 500 ns` |
| | Specify the averaging section for the $b_2$ receiver and the analyzer source port |

no. 2: Define a start value of 200 ns and a stop value of 500 ns.

```
PULS:REC:B2:SRCP2:LIN ON
```

Display the section limit lines in the diagram.

```
PULS:REC:B2:SRCP2:EVAL:MODE MEAN
```

Enable averaging of the $b_2$ wave.

```
PULS:REC:B2:SRCP2:TRIG:DEL 100 ns
```

Shift the stimulus axis of the $b_2$ wave by 100 ns.

## [SENSe<Ch>:]PULSe:RECeiver:B<rec_no>:SRCPort<port_no>:EVALuation:STARt <section_start>

Defines the start time of the averaging section for the wave quantity assigned to the measurement receiver <rec_no> and the analyzer source port <port_no>.

| | |
|---|---|
| **<Ch>** | Channel number |
| **<rec_no>** | Number of the measurement receiver (= port number of the analyzer). Values range from 1 to the total number of test ports. |
| **<port_no>** | Number of the analyzer source port. 1 to the total number of test ports. |
| **<section_start>** | Start time of the section |
| Range [def. unit] | –0.003 s to +0.002999975 s [s]. The increment (UP/DOWN) is 50 ns. The width of the averaging section must be between 25 ns and 3 s. |
| *RST value | 0 s |
| **SCPI, Command Types** | Device-specific, command or query |
| **Example:** | `See [SENSe<Ch>:]PULSe:RECeiver:B<rec_no>:SRCPort<gen_no>:EVALuation:MODE` |

## [SENSe<Ch>:]PULSe:RECeiver:B<rec_no>:SRCPort<port_no>:EVALuation:STOP <section_stop>

Defines the stop time of the averaging section for the wave quantity assigned to the measurement receiver <rec_no> and the analyzer source port <port_no>.

| | |
|---|---|
| **<Ch>** | Channel number |
| **<rec_no>** | Number of the measurement receiver (= port number of the analyzer). Values range from 1 to the total number of test ports. |
| **<port_no>** | Number of the analyzer source port. 1 to the total number of test ports. |
| **<section_stop>** | Stop time of the section |
| Range [def. unit] | –0.002999975 s to +0.003 s [s]. The increment (UP/DOWN) is 50 ns. The width of the averaging section must be between 25 ns and 3 s. |
| *RST value | 0 s |
| **SCPI, Command Types** | Device-specific, command or query |

**Example:** See [SENSe<Ch>:]PULSe:RECeiver:B<rec_no>:SRCPort<gen_no>:EVALuation:MODE

## [SENSe<Ch>:]PULSe:RECeiver:B<rec_no>:SRCPort<port_no>:LINes[:STATe] <Boolean>

Displays or hides the limit lines of the averaging section for the wave quantity assigned to the measurement receiver <rec_no> and the analyzer source port <port_no>.

| | |
|---|---|
| **<Ch>** | Channel number |
| **<rec_no>** | Number of the measurement receiver (= port number of the analyzer). Values range from 1 to the total number of test ports. |
| **<port_no>** | Number of the analyzer source port. 1 to the total number of test ports. |
| **<Boolean>** | ON \| OFF – Display or hide lines |
| *RST value | OFF |
| **SCPI, Command Types** | Device-specific, command or query |
| **Example:** | See [SENSe<Ch>:]PULSe:RECeiver:B<rec_no>:SRCPort<gen_no>:EVALuation:MODE |

## [SENSe<Ch>:]PULSe:RECeiver:B<rec_no>:SRCPort<port_no>:TRIGger:DELay <shift_stimulus>

Defines an offset time (shift stimulus) for the wave quantity assigned to the measurement receiver <rec_no> and the analyzer source port <port_no>.

| | |
|---|---|
| **<Ch>** | Channel number |
| **<rec_no>** | Number of the measurement receiver (= port number of the analyzer). Values range from 1 to the total number of test ports. |
| **<port_no>** | Number of the analyzer source port. 1 to the total number of test ports. |
| **<shift_stimulus>** | Offset time |
| Range [def. unit] | –0.003 s to +0.003 s [s]. The increment (UP/DOWN) is 50 ns. |
| *RST value | 0 s |
| **SCPI, Command Types** | Device-specific, command or query |
| **Example:** | See [SENSe<Ch>:]PULSe:RECeiver:B<rec_no>:SRCPort<gen_no>:EVALuation:MODE |

## [SENSe<Ch>:]PULSe:TIME:BWIDth[:RESolution] <bandwidth>

Selects the IF bandwidth for pulse profile measurements.

| | |
|---|---|
| **<Ch>** | Channel number |
| **<bandwidth>** | IF bandwidth |
| Range [def. unit] | 100 kHz to 10 MHz [Hz]. UP and DOWN increment/decrement the bandwidth in 1-2-5 steps for each decade (the value 3 MHz is provided in addition). The |

| | |
|---|---|
| | analyzer rounds up any entered value between these steps and rounds down values exceeding the maximum bandwidth. |
| *RST value | 10 MHz |
| **SCPI, Command Types** | Device-specific, command or query |
| **Example:** | `*RST; SWE:TYPE PULS; :PULS:TIME:BWID 5 MHz` |
| | Select pulse profile mode and select an IF bandwidth of 5 MHz. |
| | `PULS:TIME:STAR -10 us; STOP 200 us` |
| | Select a displayed time range between –10 µs and +200 µs. |

## [SENSe<Ch>:]PULSe:TIME:STARt <start>

Selects the start time of the displayed time range relative to the trigger time.

| | |
|---|---|
| **<Ch>** | Channel number |
| **<start>** | Start time |
| Range [def. unit] | –0.003 s to +0.002999975 s [s]. The increment (UP/DOWN) is 50 ns. The width of the displayed time range must be between 25 ns and 3 s. |
| *RST value | –10 µs |
| **SCPI, Command Types** | Device-specific, command or query |
| **Example:** | `See [SENSe<Ch>:]PULSe:TIME:BWIDth[:RESolution]` |

## [SENSe<Ch>:]PULSe:TIME:STOP <stop>

Selects the stop time of the displayed time range relative to the trigger time.

| | |
|---|---|
| **<Ch>** | Channel number |
| **<stop>** | Stop time |
| Range [def. unit] | –0.002999975 s to +0.003 s [s]. The increment (UP/DOWN) is 50 ns. The width of the displayed time range must be between 25 ns and 3 s. |
| *RST value | +100 µs |
| **SCPI, Command Types** | Device-specific, command or query |
| **Example:** | `See [SENSe<Ch>:]PULSe:TIME:BWIDth[:RESolution]` |

## [SENSe<Ch>:]ROSCillator...

This subsystem controls the frequency reference oscillator.

## [SENSe<Ch>:]ROSCillator[:SOURce] INTernal | EXTernal

Selects the source of the reference oscillator signal.

| | |
|---|---|
| **<Ch>** | Channel number. This suffix is ignored in the `ROSCillator` subsystem and can be set to any value. |
| **INTernal** | Select internal 10 MHz reference oscillator. |
| **EXTernal** | Select external reference clock. The frequency of the external reference clock is specified via `[SENSe<Chn>:]ROSCillator:EXTernal:FREQuency`. |
| *RST value | INTernal |
| **SCPI, Command Types** | Confirmed, command or query |
| **Example:** | `ROSC EXT` |
| | Select external reference oscillator. |
| | `ROSC:EXT:FREQ?` |
| | Query the frequency of the external reference oscillator. The response is 10000000 Hz, i.e. the frequency of the external reference oscillator must be 10 MHz. |

## [SENSe<Ch>:]ROSCillator:EXTernal:FREQuency <numeric_value>

Specifies or queries the frequency of the external reference oscillator.

| | |
|---|---|
| **<Ch>** | Channel number. This suffix is ignored in the `ROSCillator` subsystem and can be set to any value. |
| **<numeric_value>** | Frequency of the external reference clock signal. The frequency must be 10 MHz; any other setting is ignored. |
| Range [def. unit] | 10 MHz [Hz]. |
| *RST value | 10 MHz |
| **SCPI, Command Types** | Confirmed, command or query |
| **Example:** | `See [SENSe<Ch>:]ROSCillator[:SOURce].` |

## [SENSe<Ch>:]SEGMent<Seg>...

This subsystem defines all channel settings for a *Segmented Frequency* sweep. A segmented sweep is activated via `[SENSe<Ch>:]SWEep:TPYE SEGMent`.

*The commands in the `[SENSe<Ch>:]SEGMent<Seg>...` 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

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.

*Use `[SENSe<Ch>:]SEGMent<Seg>:INSert` to create a segment with specific channel settings.*

**<Ch>** Channel number. If unspecified the numeric suffix is set to 1.

**<Seg>** Sweep segment number. Segment numbers must be sequential. If n segments already exist, the added segment must have the segment number n+1.

*RST value –

**SCPI, Command Types** Device-specific, no query.

**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.

## [SENSe<Ch>:]SEGMent<Seg>:BWIDth[:RESolution] <numeric_value>

Defines the resolution bandwidth of the analyzer *(Meas. Bandwidth)* in sweep segment no. <Seg>. At the same time, the command activates separate bandwidth setting in all sweep segments (`[SENSe<Ch>:]SEGMent<Seg>:BWIDth[:RESolution]:CONTrol ON`).

**<Ch>** Channel number.

**<Seg>** Sweep segment number.

**<numeric_value>** Internal source power.

Range [def. unit] 1.0E-6 Hz to 5 MHz [Hz]. UP and DOWN increment/decrement the bandwidth in 1-2-5 steps for each decade. The analyzer rounds up any entered value between these steps and rounds down values exceeding the maximum bandwidth.

*RST value 10 kHz

**SCPI, Command Types** Device-specific, command or query

**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:BWID 1 MHZ
```

Increase the resolution bandwidth to 1 MHz.

## [SENSe<Ch>:]SEGMent<Seg>:BWIDth[:RESolution]:CONTrol <Boolean>

Qualifies whether or not the *Meas. Bandwidth* can be set independently for each sweep segment.

**<Ch>** Channel number.

**<Seg>** Sweep segment number.

**<Boolean>** ON: The bandwidth can be set independently for each sweep segment.
OFF: The bandwidth in all sweep segments is equal to the bandwidth for unsegmented sweeps set via `SENSe<Ch>:BWIDth[:RESolution]`.

*RST value OFF. The parameter is automatically switched to ON when a bandwidth is entered using `[SENSe<Ch>:]SEGMent<Seg>:BWIDth[:RESolution]`.

**SCPI, Command Types** Device-specific, command or query

**Example:**

```
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 resolution 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.

## [SENSe<Ch>:]SEGMent<Seg>:BWIDth[:RESolution]:SELect NORMal | HIGH

Defines the *Selectivity* in sweep segment no. <Seg>. At the same time, the command activates separate selectivity setting in all sweep segments (`[SENSe<Ch>:]SEGMent<Seg>:BWIDth[:RESolution]SELect:CONTrol ON`).

**<Ch>** Channel number.

**<Seg>** Sweep segment number.

**NORMal** Use IF filter with normal selectivity and short settling time.

**HIGH** Use IF filter with higher selectivity but larger settling time.

*RST value NORMal

**SCPI, Command Types** Device-specific, command or query

**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:BWID:SEL HIGH
```

Activate an IF filter with high selectivity.

## [SENSe<Ch>:]SEGMent<Seg>:BWIDth[:RESolution]:SELect:CONTrol <Boolean>

Qualifies whether or not the *Selectivity* can be set independently for each sweep segment.

**<Ch>** Channel number.

**<Seg>** Sweep segment number.

**<Boolean>** ON – The selectivity can be set independently for each sweep segment.
OFF – The selectivity in all sweep segments is equal to the selectivity for unsegmented sweeps set via
`SENSe<Ch>:BWIDth[:RESolution]:SELect`.

*RST value OFF. The parameter is automatically switched to ON when a bandwidth is entered using
`[SENSe<Ch>:]SEGMent<Seg>:BWIDth[:RESolution]:SELect`.

**SCPI, Command Types** Device-specific, command or query

**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.

## [SENSe<Ch>:]SEGMent:CLEar

Deletes all sweep segments in the channel. The command is equivalent to `[SENSe<Ch>:]SEGMent:DELet:ALL`.

## [SENSe<Ch>:]SEGMent<Seg>:COUNt?

Returns the number of sweep segments in the channel including all segments that are switched off (`[SENSe<Ch>:]SEGMent<Seg>[:STATe] OFF`).

**<Ch>** Channel number.

*RST value –

**SCPI, Command Types** Device-specific, query only.

**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.

## [SENSe<Ch>:]SEGMent<Seg>:DEFine <Start>,<Stop>,<Points>,<Power>,<Time>|<Point Delay>,<Unused>,<Meas. Bandwidth>[,<LO>,<Selectivity>]

Re-defines a sweep segment with specific channel settings *(Insert New Segment)*. The segment replaces an existing segment <Seg> in the segment list. The modified segment must not overlap with any of the existing segments.

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.*

**<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.

**<Start>, <Stop>** Start and stop frequency of the segment. See `[SENSe<Ch>:]SEGMent<Seg>:FREQuency:STARt` and `[SENSe<Ch>:]SEGMent<Seg>:FREQuency:STOP`.

Range [def. unit] A new segment must not overlap with any of the existing segments. Besides the frequency range depends on the analyzer model. [Hz]

**<Points>** Number of sweep points in the segment. See `[SENSe<Ch>:]SEGMent<Seg>:SWEep:POINts`.

Range [def. unit] 1 to 2147483647. [1] 1 is allowed if start and stop frequencies are equal.

**<Power>** Internal source power in the segment. See `[SENSe<Ch>:]SEGMent<Seg>:POWer`.

Range [def. unit] –40 dBm to +10 dBm. The exact range depends on the analyzer model; refer to the data sheet [dBm]. UP and DOWN increment/decrement the source power in 1-dB steps.

**<Time>** 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 <Point Delay>.

Range [def. unit] Depending on other channel settings [s]. `AUTO` activates automatic

| | |
|---|---|
| | sweep time setting in the segment, which is equivalent to the minimum sweep time possible. |
| **<Point Delay>** | 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 <Time>. |
| Range [def. unit] | 0 s to 2.5E+003 s. [s] `AUTO` activates automatic sweep time setting in the segment, which is equivalent to a meas. delay of 0 s. |
| ZVR **<Unused>** | Ignored parameter, should be set to the default value 0. |
| **<Meas. Bandwidth>** | Resolution bandwidth in the segment. See `[SENSe<Ch>:]SEGMent<Seg>:BWIDth[:RESolution]`. |
| Range [def. unit] | 1.0E-6 Hz to 5 MHz [Hz]. |
| **<LO>** | Position of the local oscillator frequency LO relative to the RF frequency *(Spur Avoid)*. In remote control this parameter must be set when a sweep segment is created. |
| Range [def. unit] | POSitive: LO > RF<br>NEGative: LO < RF |
| **<Selectivity>** | Selectivity of the IF filter. See `[SENSe<Ch>:]SEGMent<Seg>:BWIDth[:RESolution]:SELect`. |
| Range [def. unit] | NORMal \| HIGH [–] |
| **SCPI, Command Types** | Device-specific, with query (used to retrieve the channel settings for a particular sweep segment). |
| **Example:** | `SEGM:ADD`<br>Create a new sweep segment no. 1 in channel no. 1 using default settings.<br>`SEGM:DEF?`<br>Query the channel settings for the new segment. |

## [SENSe<Ch>:]SEGMent<Seg>:DEFine:SELect SWTime | DWELl

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 or as a meas. delay.

| | |
|---|---|
| **<Ch>** | Channel number. |
| **<Seg>** | Sweep segment number. |
| **SWTime** | Use segment sweep time. |

**DWELI** Use meas. delay.

*RST value SWTime

**SCPI, Command Types** Device-specific, command or query.

**Example:**

```
SEGM:DEF:SEL DWEL
```

Select the meas. delay to determine the sweep time in a new sweep segment.

```
SEGM: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 meas. delay of 10 ms..

```
SEGM:SWE:TIME?
```

Query the sweep time in the new segment.

## [SENSe<Ch>:]SEGMent<Seg>:DELete

Deletes the specified (single) sweep segment *(Del. Selected Segment)*. `[SENSe<Ch>:]SEGMent<Seg>:DELete:ALL` deletes all segments in the channel.

**<Ch>** Channel number.

**<Seg>** Sweep segment number. If unspecified, the value is set to 1.

*RST value –

**SCPI, Command Types** Device-specific, no query.

**Example:**

```
SEGM:ADD
```

Create a new sweep segment no. 1 in channel no. 1 using default settings and thus NORMal selectivity.

```
SEGM:DEL
```

Delete the created segment.

## [SENSe<Ch>:]SEGMent:DELete:ALL

Deletes all sweep segments in the channel *(Del. All Segments)*. `[SENSe<Ch>:]SEGMent<Seg>:DELete` deletes a single segment.

**<Ch>** Channel number.

*RST value –

**SCPI, Command Types** Device-specific, no query.

**Example:**

```
SEGM:ADD
```

Create a new sweep segment no. 1 in channel no. 1 using default settings and thus NORMal selectivity.

```
SEGM:ALL
```

Delete the created segment and all segments in the channel created before.

## [SENSe<Ch>:]SEGMent<Seg>:FREQuency:CENTer?

Returns the center frequency of sweep segment no. <Seg>.

**<Ch>** Channel number.

**<Seg>** Sweep segment number.

**Response** Center frequency of the sweep.
Range [def. unit] Depending on the instrument model [Hz].
*RST value –

**SCPI, Command Types** Device specific, query only.

**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?
```

Query the center frequency of the created segment. The response is 1.2500000000E+006.

*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:SPAN? <numeric_value>

Returns the width of the frequency range of sweep segment no. <Seg>.

**<Ch>** Channel number.

**<Seg>** Sweep segment number.

**Response** Frequency span of the sweep.
Range [def. unit] Depending on the instrument model [Hz].
*RST value –

**SCPI, Command Types** Device specific, query only.

**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:SPAN?
```

Query the span of the created segment. The response is 5.0000000000E+005.

*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 <numeric_value>

Defines the *Start* frequency of sweep segment no. <Seg>.

**<Ch>** Channel number.

**<Seg>** Sweep segment number.

**<numeric_value>** Start frequency of the sweep.

Range [def. unit] Depending on the instrument model [Hz]. The increment is 0.1 kHz.

*RST value – (no sweep segment defined after reset)

**SCPI, Command Types** Device-specific, command or query

**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?
```

Query the start frequency of the created segment. The response is 1.0000000000E+006.

***Note:*** *If the start frequency entered is greater than the current stop frequency ([SENSe<Ch>:]SEGMent<Seg>:FREQuency:STOP), the stop frequency is set to the start frequency plus the minimum frequency span of 1 Hz.*

## [SENSe<Ch>:]SEGMent<Seg>:FREQuency:STOP <numeric_value>

Defines the *Stop* frequency of sweep segment no. <Seg>.

**<Ch>** Channel number.

**<Seg>** Sweep segment number.

**<numeric_value>** Stop frequency of the sweep.

Range [def. unit] Depending on the instrument model [Hz]. The increment is 0.1 kHz.

*RST value – (no sweep segment defined after reset)

**SCPI, Command Types** Device-specific, command or query

**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:STOP?
```

Query the stop frequency of the created segment. The response is 1.5000000000E+006.

***Note:*** *If the stop frequency entered is smaller than the current start frequency ([SENSe<Ch>:]SEGMent<Seg>:FREQuency:STARt), the start frequency is set to the stop frequency minus the minimum frequency span ([SENSe<Ch>:]SEGMent<Seg>:FREQuency:SPAN).*

## [SENSe<Ch>:]SEGMent<Seg>:INSert <Start>,<Stop>,<Points>,<Power>,<Time>|<Point Delay>,<Unused>,<Meas. Bandwidth>[,<LO>,<Selectivity>,<IF Gain>]

Adds a new sweep segment with specific channel settings. The new segment must not overlap with any of the existing segments.

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.*

**<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. Moreover, segment numbers must selected such that the corresponding frequency ranges are in ascending order.
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>.

**<Start>, <Stop>** Start and stop frequency of the segment. See `[SENSe<Ch>:]SEGMent<Seg>:FREQuency:STARt` and `[SENSe<Ch>:]SEGMent<Seg>:FREQuency:STOP`.

Range [def. unit] A new segment must not overlap with any of the existing segments. Besides the frequency range depends on the analyzer model. [Hz]

**<Points>** Number of sweep points in the segment. See `[SENSe<Ch>:]SEGMent<Seg>:SWEep:POINts`.

Range [def. unit] 1 to 2147483647. [1] 1 is allowed if start and stop frequencies are equal.

**<Power>** Internal source power in the segment. See `[SENSe<Ch>:]SEGMent<Seg>:POWer`.

Range [def. unit] –40 dBm to +10 dBm. The exact range depends on the analyzer model; refer to the data sheet [dBm]. UP and DOWN increment/decrement the source power in 1-dB steps.

**<Time>** 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 <Point Delay>. `AUTO` activates automatic sweep time setting in the segment, which is equivalent to the minimum sweep time possible. |
| Range [def. unit] | Depending on other channel settings [s]. |
| **<Point Delay>** | Delay for each partial measurement in the segment. See `[SENSe<Ch>:]SEGMent<Seg>:SWEep:DWELl`. In the setting `[SENSe<Ch>:]SEGMent<Seg>:INSert:SELec SWTime`, this parameter is replaced by <Time>. |
| Range [def. unit] | 0 s to 2.5E+003 s. [s] `AUTO` activates automatic sweep time setting in the segment, which is equivalent to a meas. delay of 0 s. |
| ZVR ☑**<Unused>** | Ignored parameter, should be set to the default value 0. |
| **<Meas. Bandwidth>** | Resolution bandwidth in the segment. See `[SENSe<Ch>:]SEGMent<Seg>:BWIDth[:RESolution]`. |
| Range [def. unit] | 1.0E-6 Hz to 5 MHz [Hz]. |
| **<LO>** | Position of the local oscillator frequency LO relative to the RF frequency *(Spur Avoid)*. In remote control this parameter must be set when a sweep segment is created. |
| Range [def. unit] | POSitive: LO > RF<br>NEGative: LO < RF |
| **<Selectivity>** | Selectivity of the IF filter. See `[SENSe<Ch>:]SEGMent<Seg>:BWIDth[:RESolution]:SELect`. |
| Range [def. unit] | NORMal \| HIGH [–] |
| **<IF Gain>** | IF gain in the receiver path, valid for all ports. |
| Range [def. unit] | LNOise: Low Noise, for high dynamic range (low input levels)<br>LDIStortion: Low distortion, for high overdrive reserve (high input levels)<br>AUTO: Automatic selection of IF gain according to the received signal level (Automatic Gain Control, AGC)<br>CHANnel: For future extensions |
| **SCPI, Command Types** | Device-specific, no query. |
| **Example:** | `SEGM:INS 1MHZ, 1.5MHZ, 111, -21DBM, 0.5S, 0, 10KHZ`<br>Create a sweep segment with a sweep range between 1.0 MHz and 1.5 MHz.<br>`SEGM2:ADD`<br>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. |

## [SENSe<Ch>:]SEGMent<Seg>:INSert:SELect SWTime | DWELl

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 or as a meas. delay.

**<Ch>** Channel number.

**<Seg>** Sweep segment number.

**SWTime** Use segment sweep time.

**DWELl** Use meas delay.

*RST value SWTime

**SCPI, Command Types** Device-specific, command or query

**Example:**

```
SEGM:INS:SEL DWEL
```

Select the meas. delay to determine the sweep time in a new sweep segment.

```
SEGM: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..

```
SEGM:SWE:TIME?
```

Query the sweep time in the new segment.

## [SENSe<Ch>:]SEGMent<Seg>:OVERlap <Boolean>

Queries whether the analyzer supports overlapping sweep segments.

**<Ch>** Channel number.

**<Seg>** Sweep segment number.

**<Boolean>** ON | OFF: No effect.

*RST value OFF. If used as a query, the command returns the information that overlapping sweep segments are not supported *(OFF)*.

**SCPI, Command Types** Device-specific, command (no effect) or query

## [SENSe<Ch>:]SEGMent<Seg>:POWer[:LEVel] <numeric_value>

Defines the *Power* of the internal signal source in sweep segment no. <Seg>. At the same time, the command activates separate power control in all sweep segments (`[SENSe<Ch>:]SEGMent<Seg>:POWer[:Level]:CONTrol ON`).

**<Ch>** Channel number.

**<Seg>** Sweep segment number.

**<numeric_value>** Internal source power.

Range [def. unit] –40 dBm to +10 dBm. The exact range depends on the analyzer model; refer to the data sheet [dBm].

*RST value –10 dBm.

**SCPI, Command Types** Device-specific, command or query

**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.

## [SENSe<Ch>:]SEGMent<Seg>:POWer[:LEVel]:CONTrol <Boolean>

Qualifies whether or not the *Power* of the internal signal source can be set independently for each sweep segment.

**<Ch>** Channel number.

**<Seg>** Sweep segment number.

**<Boolean>** ON: The power can be set independently for each sweep segment.
OFF: The power in all sweep segments is equal to the internal source power for unsegmented sweeps set via
`SOURce<Ch>:POWer[:LEVel][:IMMediate][:AMPlitude]`.

*RST value OFF. The parameter is automatically switched to ON when a segment level is entered using
`[SENSe<Ch>:]SEGMent<Seg>:POWer[:LEVel]`.

**SCPI, Command Types** Device-specific, command or query

**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.

## [SENSe<Ch>:]SEGMent<Seg>[:STATe] <Boolean>

Activates or deactivates the sweep segment <Seg>. Sweep points belonging to inactive segments only are not measured

**<Ch>** Channel number.

**<Seg>** Sweep segment number.

**<Boolean>** ON | OFF - Activates or deactivates the measurement in sweep segment <Seg>.

*RST value ON

**SCPI, Command Types** Device-specific, command or query

**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.

## [SENSe<Ch>:]SEGMent<Seg>:SWEep:DWELl <numeric_value>

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.

**<Ch>** Channel number.

**<Seg>** Sweep segment number.

**<numeric_value>** Meas. delay before each partial measurement.

Range [def. unit] 0 s to 2.5E+003 s. [s] 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).

*RST value 0 s

**SCPI, Command Types** Device-specific, command or query. This default value corresponds to automatic sweep time setting in manual control.

**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.

## [SENSe<Ch>:]SEGMent<Seg>:SWEep:DWELl:CONTrol <Boolean>

Qualifies whether or not the *Meas. Delay* defined via `[SENSe<Ch>:]SEGMent<Seg>:SWEep:DWELl` can be set independently for each sweep segment.

**<Ch>** Channel number.

**<Seg>** Sweep segment number.

**<Boolean>** ON: The meas. delay can be set independently for each sweep segment.
OFF: The meas. delay in all sweep segments is equal to the *Meas.*

*Delay* measurements for unsegmented sweeps set via `[SENSe<Ch>:]SWEep:DWELl`.

*RST value OFF. The parameter is automatically switched to ON when a meas. delay time is entered using `[SENSe<Ch>:]SEGMent<Seg>:SWEep:DWELl`.

**SCPI, Command Types** Device-specific, command or query.

**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.

## [SENSe<Ch>:]SEGMent<Seg>:SWEep:POINts <numeric_value>

Defines the total number of measurement *Points* in sweep segment no. <Seg>.

**<Ch>** Channel number.

**<Seg>** Sweep segment number.

**<numeric_value>** Number of points in the segment.

Range [def. unit] 1 to 2147483647. [1] 1 is allowed if start and stop frequencies are equal.

*RST value 51

**SCPI, Command Types** Device-specific, command or query

**Example:**

```
SEGM:ADD
```

Create a new sweep segment no. 1 in channel no. 1 using

default settings and thus 51 sweep points.

```
SEGM:SWE:POIN 401
```

Increase the number of points to 401.

## [SENSe<Ch>:]SEGMent<Seg>:SWEep:TIME <numeric_value>

Sets the duration of the sweep in sweep segment no. <Seg> *(Segment Sweep Time)*. At the same time, the command activates separate sweep time setting in all sweep segments (`[SENSe<Ch>:]SEGMent<Seg>:SWEep:TIME:CONTrol ON`).

**<Ch>** Channel number.

**<Seg>** Sweep segment number.

**<numeric_value>** Internal source power.

Range [def. unit] The minimum duration depends on the other channel settings, in particular on the number of points (`[SENSe<Ch>:]SEGMent<Seg>:SWEep:POINts`), the IF bandwidth (BWID) and the delay for each partial measurement (`[SENSe<Ch>:]SEGMent<Seg>:SWEep:DWELl`). The maximum is 1000 s. [s] Changing the duration leaves the number of points unchanged but directly affects the delay.

*RST value MIN, depending on the channel settings. This default value corresponds to automatic sweep time setting in manual control.

**SCPI, Command Types** Device-specific, command or query

**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:SWE:TIME 0.1
```

Increase the segment sweep time to 0.1 s.

## [SENSe<Ch>:]SEGMent<Seg>:SWEep:TIME:CONTrol <Boolean>

Qualifies whether or not the *Segment Sweep Time* can be set independently for each sweep segment.

| | |
|---|---|
| **<Ch>** | Channel number. |
| **<Seg>** | Sweep segment number. |
| **<Boolean>** | ON: The time can be set independently for each sweep segment.<br>OFF: The time in all sweep segments is equal to the sweep time for unsegmented sweeps set via `[SENSe<Ch>:]SWEep:TIME`. |
| *RST value | OFF. 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. |
| **SCPI, Command Types** | Device-specific, command or query |

**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: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.

## [SENSe<Ch>:]SWEep...

This subsystem provides general settings to control the sweep . Most of the settings are related to the sweep time.

NOTE *The [SENSe<Ch>:]SWEep... settings apply to non-segmented sweep types. Segmented sweeps are selected via `[SENSe<Ch>:]SWEep:TPYE SEGMent` and configured in the [SENSe<Ch>:]SEGMent<Seg>... subsystem.*

## [SENSe<Ch>:]SWEep:AXIS:FREQuency '<frequency_ref>'

Selects the stimulus signal that is used for the definition of the frequency sweep range or CW

frequency.

This command is particularly useful in arbitrary mode (with option R&S ZVA-K4), where the frequencies and source powers at the ports are independent. For R&S ZVA analyzers without option R&S ZVA-K4, all frequencies are coupled so that this selection has no effect.

**<Ch>** Channel number. If unspecified the numeric suffix is set to 1.

**'<frequency_ref>'** 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
...
'Port All; Receiver' – Receiver frequency (equal for all analyzer ports)
'Gen 1; Source' – External generator frequency 1
...
'Pmtr 1; Receiver' – External power meter frequency 1
...

*RST value 'Channel Base: Source'

**SCPI, Command Types** Device-specific, command or query

**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.

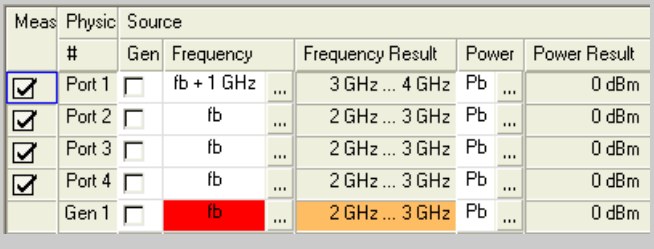
```
SWE:AXIS:FREQ 'Port 2; Source'
```

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`.

```
DISP:MENU:KEY:EXEC 'Port Config'
```



## [SENSe<Ch>:]SWEep:AXIS:POWer '<power_ref>'

Selects the stimulus signal that is used for the definition of the power sweep range or fixed power.

This command is particularly useful in arbitrary mode (with option R&S ZVA-K4), where the frequencies and source powers at the ports are independent. For R&S ZVA analyzers without option R&S ZVA-K4, all powers are coupled so that this selection has no effect.

**<Ch>** Channel number.

**'<frequency_ref>'** String parameter, one of the following signals:
'Channel Base; Source' – Base channel power
'Port 1; Source' – Source power at port 1,
'Port 2; Source' – Source power at port 2
...
'Gen 1; Source' – External generator power 1
...

*RST value 'Channel Base; Source'

**SCPI, Command Types** Device-specific, command or query

**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.

```
DISP:MENU:KEY:EXEC 'Port Config'
```

| Meas | Physic | Source | | | | | |
|---|---|---|---|---|---|---|---|
| | # | Gen | Frequency | Frequency Result | Power | | Power Result |
| ☑ | Port 1 | ☐ | fb ... | 10 MHz ... 400 GHz | Pb - 5 dB | ... | -5 dBm |
| ☑ | Port 2 | ☐ | fb ... | 10 MHz ... 400 GHz | Pb | ... | 0 dBm |
| ☑ | Port 3 | ☐ | fb ... | 10 MHz ... 400 GHz | Pb | ... | 0 dBm |
| ☑ | Port 4 | ☐ | fb ... | 10 MHz ... 400 GHz | Pb | ... | 0 dBm |
| | Gen 1 | ☐ | fb ... | 10 MHz ... 400 GHz | Pb | ... | 0 dBm |

## [SENSe<Ch>:]SWEep:COUNt <No_of_Sweeps>

Defines the number of sweeps to be measured in single sweep mode (`INITiate<Ch>:CONTinuous`

OFF).

**<Ch>** Channel number.

**<No_of_Sweeps>** Number of consecutive sweeps to be measured.
Range [def. unit] 1 to 999. [–]
*RST value 1

**SCPI, Command Types** Confirmed, command or query

**Example:**

```
See CALCulate<Chn>:DATA:NSWeep?.
```

## [SENSe<Ch>:]SWEep:DETector:TIME <det_time>

Defines the detector observation time ratios and wave quantities if a *Peak* or *RMS* detector is active. The detector is selected together with the measured quantity (CALCulate<Ch>:PARameter:MEASure or CALCulate<Ch>:PARameter:SDEFine).

**<Ch>** Channel number.

**<det_time>** Detector time.
Range [def. unit] 0 s to 3456000 s. [s]
*RST value 0.01 s

**SCPI, Command Types** Confirmed, command or query

**Example:**

```
*RST; CALC:PAR:MEAS 'TRC1', 'a1D1RMS'
```

Select the wave quantity a1 for the default trace and activate the RMS detector.

```
SWE:DET:TIME 1
```

Specify an observation time of 1 s at each sweep points.

## [SENSe<Ch>:]SWEep:DWELl <delay>

Defines the *Meas. Delay* time for each partial measurement. Setting a delay disables the automatic calculation of the (minimum) sweep time; see [SENSe<Ch>:]SWEep:TIME:AUTO.

**<Ch>** Channel number.

**<delay>** Meas. delay before each partial measurement.
Range [def. unit] 0 s to 3.179551E+002 s. [s] Changing the delay leaves the number of points unchanged but has an impact on the duration of the sweep ([SENSe<Ch>:]SWEep:TIME).
*RST value 0 s

**SCPI, Command Types** Confirmed, command or query

**Example:**

```
FUNC "XFR:POW:S12"
```

Activate a frequency sweep and select the S-parameter S12 as measured parameter for channel and trace no. 1.

```
SWE:TIME?
```

Query total sweep time.

```
SWE:DWEL 1
```

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 partial measurements per sweep.

## [SENSe<Ch>:]SWEep:POINts <no_points>

Defines the total number of measurement points per sweep *(Number of Points)*.

**<Ch>** Channel number 1.

**<no_points>** Number of points per sweep.
Range [def. unit] 2 to 2147483647. [1]
*RST value 201

**SCPI, Command Types** Confirmed, command or query

**Example:**

```
FUNC "XFR:POW:S12"
```

Activate a frequency sweep and select the S-parameter S12 as measured parameter for channel and trace no. 1.

```
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.

## [SENSe<Ch>:]SWEep:SPACing LINear | LOGarithmic

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:TPYE` to select sweep types other than Lin. Frequency or Log. Frequency.*

**<Ch>** Channel number.

**LINear** The stimulus frequency is swept in equidistant steps over the frequency range. In a Cartesian diagram, the x-axis is a linear frequency axis.

**LOGarithmic** The frequency is swept in equidistant steps on a logarithmic scale. In a Cartesian diagram, the x-axis is a logarithmic frequency axis.

*RST value LINear

**SCPI, Command Types** Confirmed, command or query

**Example:**

```
FUNC "XFR:POW:S12"
```

Activate a frequency sweep and select the S-parameter S12 as measured parameter for channel and trace no. 1.

```
SWE:SPAC LOG
```

Change to sweep type *Log Frequency*.

## [SENSe<Chn>:]SWEep:SRCPort 1 | 2 | 3 | 4

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:

**<Chn>** Channel number used to identify the active trace. If unspecified the numeric suffix is set to 1.

**1 ...** Test port number of the analyzer, 1 to 2, 3 or 4, depending on the analyzer model.

*RST value 1 (test port 1)

**SCPI, Command Types** Device-specific, with query.

**Example:**

```
CALC4:PAR:SDEF "Ch4Tr1", "A1"
```

Create channel 4 and a trace named *Ch4Tr1* to measure the wave quantity a1. The trace automatically becomes the active trace.

```
SENS4:SWE:SRCP 2
```

Select drive port 2 for the active trace.

## [SENSe<Ch>:]SWEep:STEP <step_size>

Sets the distance between two consecutive sweep points.

*This setting is valid for sweep types with equidistant sweep points only. It does not apply to logarithmic and segmented sweeps.*

**<Ch>** Channel number.

**<step_size>** Stimulus step size.

Range [def. unit] Depending on the other channel settings. The minimum step size is equal to the sweep span divided by the maximum number of points minus one ([SENSe<Ch>:]SWEep:POINts), the maximum step size is equal to the sweep span ([SENSe<Ch>:]FREQuency:SPAN). See also description of manual control and program example below. [–]

*RST value Depending on the analyzer model. The default step size is equal to the default sweep span of the analyzer divided by the default number of

points minus one.

**SCPI, Command Types** Confirmed, with query.

**Example:**

```
*RST; SWE:STEP?
```

Query the default step size. For a 24 GHz analyzer, the response is 119950000. 119950000 Hz = (24 GHz – 10 MHz) / 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.

## [SENSe<Ch>:]SWEep:TIME <duration>

Sets the duration of the sweep *(Sweep Time)*. Setting a duration disables the automatic calculation of the (minimum) sweep time; see [SENSe<Ch>:]SWEep:TIME:AUTO.

***Note:*** *The sweep duration is ignored for the sweep types Time and CW Mode.*

**<Ch>** Channel number.

**<duration>** Duration of the sweep.

Range [def. unit] The minimum duration depends on the other channel settings, in particular on the number of points ([SENSe<Ch>:]SWEep:POINts), the IF bandwidth (BWID) and the *Meas. Delay* for each partial measurement ([SENSe<Ch>:]SWEep:DWELl). The maximum is 100000 s. [s]
Changing the duration leaves the number of points unchanged but directly affects the delay.

*RST value MIN, depending on the channel settings

**SCPI, Command Types** Confirmed, command or query

**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.

## [SENSe<Ch>:]SWEep:TIME:AUTO <Boolean>

When enabled, the (minimum) sweep duration 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.*

| | |
|---|---|
| **<Ch>** | Channel number. |
| **<Boolean>** | ON \| OFF - Turns 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 value | ON |
| **SCPI, Command Types** | Confirmed, command or query |
| **Example:** | `SWE:TIME 1`<br>Set a total sweep time of 1 s.<br>`SWE:TIME:AUTO?`<br>A query returns the value 1. |

## [SENSe<Ch>:]SWEep:TYPE LINear | LOGarithmic | SEGMent | POWer | CW | POINt | PULSe | IAMPlitude | IPHase

Selects the sweep type, i.e. the sweep variable (frequency/power/time) and the position of the sweep points across the sweep range.

| | |
|---|---|
| **<Ch>** | Channel number. |
| **LINear** | Lin. frequency sweep at constant source power (`SOURce<Ch>:POWer<Pt>[: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** | Log. 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. |
| **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. |
| **POWer** | Power sweep. The measurement is performed at constant frequency (`SOURce<Ch>:FREQuency<Pt>:CW\|FIXed`) but with variable generator power that is swept in linear, equidistant steps over a continuous range (`SOURce<Ch>:POWer<Pt>:STARt`, `SOURce<Ch>:POWer<Pt>: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<Pt>:CW\|FIXed`) and source power (`SOURce<Ch>:POWer<Pt>[: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. |
| **PULSe** | Pulse profile mode, configured via `SENSe:PULSe...` |
| **IAMPlitude** | Amplitude imbalance sweep, configured via `SOURce:TDIF...` |

**IPHase** Phase imbalance sweep, configured via `SOURce:TDIF...`

*RST value LINear

**SCPI, Command Types** Device-specific, command or query

**Example:**

```
FUNC "XFR:POW:S12"
```

Activate a frequency sweep and select the S-parameter S12 as measured parameter for channel and trace no. 1.

```
SWE:TYPE LOG
```

Change to sweep type *Log Frequency*.

# SOURce<Ch>:...

This subsystem controls the frequency and power of the internal signal source, configures and starts the source power calibration, and provides output port settings.

***Note:*** *The SOURce<Ch>:POWer... subsystem comprises port-specific and general settings. Port-specific settings are valid for the port specified by the numeric suffix `<Pt>` (...`:POWER<Pt>:`...). General settings are valid for all test ports of the analyzer; the port suffix is ignored. Refer to the description of the individual commands for more information.*

## SOURce<Ch>:FREQuency<Pt>[:CW] <fixed frequency>
## SOURce<Ch>:FREQuency<Pt>:FIXed <fixed frequency>

Defines the fixed (Continuous Wave, CW) frequency for all sweep types operating at fixed frequency (power sweep, time sweep, CW mode sweep). The two command forms `SOUR:FREQ:CW` and `SOUR:FREQ:FIX` are equivalent.

NOTE `SOURce<Ch>:FREQuency<Pt>:CW|FIXed` is equivalent to `[SENSe<Ch>:]FREQuency:CW|FIXed`. Source and receiver frequency are equal; the four commands overwrite each other. With option ZVA-K4, *Frequency Conversion,* independent offset frequencies can be defined for the source and receiver frequencies; see `SOURce<Ch>:FREQuency<Pt>:CONVersion:ARBitrary:IFRequency` and `[SENSe<Ch>:]FREQuency:CONVersion:ARBitrary`.

**<Ch>** Channel number. If unspecified the numeric suffix is set to 1.

**<Pt>** Test port number of the analyzer. This suffix is ignored because the selected frequency applies to all source ports used in the active channel.

**<fixed_frequency>** Fixed stimulus and analyzer frequency.

Range [def. unit] Depending on the instrument model [Hz]. The increment (parameters UP or DOWN) is 0.1 kHz.

*RST value 1 GHz

**SCPI, Command Types** Confirmed, command or query

**Example:**

```
FUNC "XPOW:POW:A1"
```

Activate a power sweep and select the wave quantity a1 as measured parameter for channel and trace no. 1.

```
SOUR:FREQ:CW 100 MHz
```

Set the CW frequency to 100 MHz.

## SOURce<Ch>:FREQuency<Pt>:CONVersion:ARBitrary:EFRequency<Gen> <Boolean>, <numerator>, <denominator>, <offset>, CW | FIXed | SWEep

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).

**<Ch>** Channel number.

**<Pt>** Test port number of the analyzer. This suffix is ignored because the the generator is referenced via <Gen>.

**<Gen>** Generator number

**<Boolean>** Switch the generator on or off.

**<numerator>**
**<denominator>**
**<offset>**

Parameters of the frequency formula. The source frequency $f_s$ is calculated according to
$f_s$ = <numerator>/<denominator>*$f_b$. + <offset>
where $f_b$ represents the sweep range (for frequency sweeps, parameter SWEep). For power, time and *CW Mode* sweeps (parameters CW or FIXed), $f_b$ = 0.

NOTE *The conversion formula for $f_s$ is ZVR-compatible but differs from manual control where the ratio <numerator>/<denominator> also multiplies the <offset>. In manual control, the <offset> is defined as a product of a ratio of two integers times a multiple of 1 Hz.*

Range [def. unit]

The converted frequency or frequency range must be within the external generator's frequency range [Hz]. In addition the following restrictions apply:
<numerator> – Values are rounded to positive or negative integer numbers; zero is not allowed [1]
<denominator> – Values are rounded to positive integer numbers [1]
<offset> – Values are rounded to (positive or negative) multiples of 1 Hz [Hz].

*RST value

The default frequency/frequency range corresponds to the sweep range or CW frequency of the analyzer, i.e.
<numerator> = <denominator> = 1
<offset> = 0 Hz

**Sweep type**
**Parameters**

SWEep – A frequency sweep is active, the command defines a frequency range.
CW | FIXed – A power, time or *CW Mode* sweeps is active, the command defines a fixed frequency.

*RST value

SWEep

**SCPI, Command Types**

Device-specific, command or query

**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, 2, 1, 1E+9, SWE
```

Switch the generator on and convert the generator frequency to the range between 3 GHz and 3.2 GHz.

## SOURce<Ch>:FREQuency<Pt>:CONVersion:ARBitrary:IFRequency <numerator>, <denominator>, <offset>, CW | FIXed | SWEep

Defines the port-specific source frequency for frequency-converting measurements. The source frequency is either a range (for frequency sweeps) or a CW frequency (for power, time and *CW Mode* sweeps).

*The frequency formula is applied even if the analyzer returns an error message, because the frequency is outside the allowed range.*

**<Ch>** Channel number.

**<Pt>** Test port number of the analyzer. Note that the source frequencies at ports 1/2 and at ports 3/4 must be the same if the RF signal is simultaneously fed to two coupled ports.

**<numerator>**
**<denominator>**
**<offset>**

Parameters of the frequency formula. The source frequency $f_s$ is calculated according to
$f_s$ = <numerator>/<denominator>*$f_b$. + <offset>
where $f_b$ represents the sweep range (for frequency sweeps, parameter SWEep). For power, time and *CW Mode* sweeps (parameters CW or FIXed), $f_b$ = 0.

*The conversion formula for $f_s$ is ZVR-compatible but differs from manual control where the ratio <numerator>/<denominator> also multiplies the <offset>. In manual control, the <offset> is defined as a product of a ratio of two integers times a multiple of 1 Hz.*

Range [def. unit] The converted frequency or frequency range must be within the analyzer's frequency range, depending on the instrument model [Hz]. In addition the following restrictions apply:
<numerator> – Values are rounded to positive or negative integer numbers; zero is not allowed [1]
<denominator> – Values are rounded to positive integer numbers [1]
<offset> – Values are rounded to (positive or negative) multiples of 1 Hz [Hz].

*RST value The default frequency/frequency range corresponds to the sweep range or CW frequency of the analyzer, i.e.
<numerator> = <denominator> = 1
<offset> = 0 Hz

**Sweep type Parameters** SWEep – A frequency sweep is active, the command defines a frequency range.
CW | FIXed – A power, time or *CW Mode* sweeps is active, the command defines a fixed frequency.

*RST value SWEep

**SCPI, Command** Device-specific, command or query

**Types**

**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.

## SOURce<Ch>:]FREQuency:CONVersion:MIXer:FUNDamental RF | LO

Selects the mixer input signal which is at the fundamental power (channel power, to be defined by means of `SOURce<Ch>:POWer<Pt>:STARt`, `SOURce<Ch>:POWer<Pt>:STOP`, `SOURce<Ch>:POWer<Pt>[:LEVel][:IMMediate][:AMPlitude]`).

**<Ch>** Channel number.

**Parameters** RF – mixer input signal
LO – local oscillator signal

*RST value RF

**SCPI, Command Types** Device-specific, command or query.

**Example:**

```
See SOURce<Ch>:FREQuency:CONVersion:MIXer:PFIXed
```

## SOURce<Ch>:FREQuency:CONVersion:MIXer:PFIXed <fixed_power>

Defines a fixed power, to be assigned either to the RF or to the LO signal (the signal not addressed by `SOURce<Ch>:]FREQuency:CONVersion:MIXer:FUDamental RF | LO`).

**<Ch>** Channel number.

**<fixed_power>** Fixed power.

Range [def. unit] –40 dBm to +10 dBm. The exact range depends on the analyzer model or external generator used; refer to the data sheet [dBm]. UP and DOWN increment/decrement the source power in 0.1-dB steps.

*RST value –25 dBm

**SCPI, Command Types** Device-specific, command or query.

**Example:**

```
*RST; SOUR:FREQ:CONV:MIX:PFIX –10
```

Reset the analyzer and specify a fixed power of –10 dBm.

```
SOUR:FREQ:CONV:MIX:FUND LO
```

Assign the fundamental frequency to the LO signal. The fixed frequency is automatically assigned to the RF signal.

## SOURce<Ch>:GROup<group_no> <log_port1>,<log_port2>

Defines a group of measured ports numbered <group_no> containing a continuous range of logical

ports <log_port1> and <log_port2>.

**<Ch>** Channel number.

**<group_no>** Port group number. If unspecified the analyzer automatically selects an unused port group number (existing port groups are not overwritten).

Range [def. unit] Port groups must contain at least one logical port and must not overlap, so the maximum number of port groups for an n-port analyzer is n. [–]

**<log_port1>, <log_port2>** First and last logical port number in the port group. The port group contains all port with numbers between <log_port1> and <log_port2>. The command generates an error message if only one parameter is transferred. Both parameters must be omitted if the command is used as a query.

Range [def. unit] 1 to n where n is the number of ports of the analyzer. <log_port1> can be smaller than, equal to or larger than <log_port2>, however, a physical port cannot be assigned to several port groups. [–]

**SCPI, Command Types** Device-specific, command or query

**Example:**

```
SOUR:GRO1 1,2; SOUR:GRO2 3,4
```

On a 4-port analyzer, arrange the logical ports no. 1 and 2 in port group 1, the logical ports no 3 and 4 in port group 2.

```
SOUR:GRO?
```

Query the port group no. 1. The response is *1,2.*

```
SOUR:GRO:CLE ALL; SOUR:GRO1?
```

Dissolve all port group and repeat the query. The response is *1,4:* the default port group no. 1 contains all ports.

## SOURce<Ch>:GROup<group_no>:CLEar [ALL]

Dissolves a group of measured ports <group_no> or all port groups.

**<Ch>** Channel number.

**<group_no>** Port group number. If unspecified the numeric suffix is set to 1.

Range [def. unit] Range according to the current port groups (`SOURce<Ch>:GROup<group_no>`). If an undefined port group number is used, the analyzer generates an error message. Exception: Parameter `ALL`, see below. [–]

*Note: Port group no 1 is always present and can not be dissolved. After a *RST or after `SOURce<Ch>:GROup<group_no>:CLEar ALL`, port group no. 1 contains all ports of the analyzer.*

**ALL** If `ALL` is specified, all port groups are dissolved (exception: port group 1; see above); the <group_no> suffix is ignored. If `ALL` is omitted, only the specified port group is dissolved.

**SCPI, Command Types** Device-specific, no query

**Example:**

```
See SOURce<Ch>:GROup<group no>
```

## SOURce<Ch>:LPORt<log_port> <phys_port1>,<phys_port2>

Defines a balanced port numbered <log_port>, combining two physical ports <phys_port1> and <phys_port2>.

*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<log_port>` is used.*

**<Ch>** Channel number.

**<log_port>** Logical port number used to number balanced or physical ports. If unspecified the numeric suffix is set to 1.

Range [def. unit] One balanced port requires two physical ports, so the maximum number of balanced ports for an n-port analyzer is n/2 (n even) or (n – 1)/2 (n odd). If the command is used as a query, then <log_port> can range up to n (no balanced ports, see example below). [–]

**<phys_port1>, <phys_port2>** Physical port numbers. The command has no effect if only one parameter is transferred. Both parameters must be omitted if the command is used as a query.

Range [def. unit] 1 to n where n is the number of ports of the analyzer. The port numbers must be different from each other. Moreover, a physical port cannot be assigned to several physical ports. [–]

**SCPI, Command Types** Device-specific, command or query

**Example:**

```
*RST; SOUR:LPOR1 1,2; LPOR2 3,4
```

Assuming that the analyzer provides 4 test ports, combine the physical ports 1 and 2 to the logical port 1, the physical ports 3 and 4 to the logical port 2.

```
SOUR:LPOR1?
```

Query the physical ports assigned to logical port no. 1. The response is *1,2.*

```
SOUR:LPOR:CLE ALL; :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.

## SOURce<Ch>:LPORt<log_port>:CLEar [ALL]

Dissolves a balanced port numbered <log_port> or all balanced ports.

**<Ch>** Channel number.

**<log_port>** Logical port number used to number balanced ports. If unspecified the numeric suffix is set to 1.

Range [def. Range according to the current port configuration

| | |
|---|---|
| unit] | (SOURce<Ch>:LPORt<log_port>). If an undefined balanced port number is used, the analyzer generates an error message. Exception: Parameter ALL, see below. [–] |
| **ALL** | If ALL is specified, all balanced ports are dissolved; the <log_port> suffix is ignored. If ALL is omitted, only the specified balanced port is dissolved. |
| **SCPI, Command Types** | Device-specific, no query |
| **Example:** | See SOURce<Ch>:LPORt<log_port> |

## SOURce<Ch>:POWer...

This subsystem controls the power of the internal signal source and provides output port settings.

***Note:*** *The SOURce<Ch>:POWer... subsystem comprises port-specific and general settings. Port-specific settings are valid for the port specified by the numeric suffix <Pt> (...:POWER<Pt>:...). General settings are valid for all test ports of the analyzer; the port suffix is ignored. Refer to the description of the individual commands for more information.*

### SOURce<Ch>:POWer<Pt>:ATTenuation <numeric_value>

Sets a fixed attenuation factor for the generated wave at test port no. <Pt>. The incident wave is attenuated via [SENSe<Ch>:]POWer:ATTenuation.

*Use SOURce<Ch>:POWer<Pt>:ATTenuation:AUTO ON for automatic generator attenuator setting. Automatic attenuator setting overwrites the manual attenuation factor. Conversely, a manual entry resets the mode to manual (OFF).*

| | |
|---|---|
| **<Ch>** | Channel number. |
| **<Pt>** | Test port number of the analyzer. |
| **<numeric_value>** | Attenuation factor for the generated wave. |
| Range [def. unit] | Depending on the analyzer/attenuator model, e.g. 0 dB to 80 dB [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 value | 0 dB |
| **SCPI, Command Types** | Confirmed, command or query |
| **Example:** | SOUR:POW -6<br>Set the internal source power for channel 1 to -6 dBm.<br>SOUR:POW2:ATT 10<br>Set the step attenuator for the wave transmitted at port 2 to 10 dB. The internal source power and the waves at the other test ports are not affected.<br>SOUR:POW2:ATT:AUTO? |

Query whether an automatic or manual generator attenuation is used. The response is *OFF.*

```
SOUR:POW2:ATT:AUTO ON
```

Switch back to automatic attenuator setting.

## SOURce<Ch>:POWer<Pt>:ATTenuation:AUTO ON| OFF

Qualifies whether the attenuation factor for the generated wave at test port no. <Pt> is set manually or automatically.

*Use SOURce<Ch>:POWer<Pt>:ATTenuation to define a fixed generator attenuation factor. Use `SOURce<Ch>:POWer<Pt>:ATTenuation:AUTO ON` to switch back to automatic attenuator setting.*

| | |
|---|---|
| **<Ch>** | Channel number. |
| **<Pt>** | Test port number of the analyzer. |
| **Parameters** | ON – automatic generator attenuator setting<br>OFF – manual entry of a fixed attenuation step |
| *RST value | AUTO |
| **SCPI, Command Types** | Confirmed, command or query |
| **Example:** | `See SOURce<Ch>:POWer<Pt>:ATTenuation` |

## SOURce<Ch>:POWer<Pt>:GENerator<Gen>:OFFSet <numeric_value>, ONLY | CPADd

Defines the power of an external generator or its power offset relative to the channel power (`SOURce<Ch>:POWer<Pt>[:LEVel][:IMMediate][:AMPlitude]`).

| | |
|---|---|
| **<Ch>** | Channel number. |
| **<Pt>** | Test port number of the analyzer. This suffix is ignored; the generator is selected via <Gen>. |
| **<Gen>** | Generator number |
| **<numeric_value>** | Port-specific power offset. |
| Range [def. unit] | –300 dB to +300 dB [dB]. UP and DOWN increment/decrement the phase in 1-dB steps. |
| *RST value | 0 dB |
| **Offset parameters** | ONLY – only the port-specific power is used; the channel power is ignored.<br>CPADd – the port-specific power is added as an offset to the channel power. |
| *RST value | CPADd |
| **SCPI, Command Types** | Device-specific, command or query |

**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.

## SOURce<Ch>:POWer<Pt>:GENerator<Gen>:PERManent[:STATe] <Boolean>

Defines whether the external generator power is permanently on.

**<Ch>** Channel number.

**<Pt>** Test port number of the analyzer. This suffix is ignored; the generator is selected via <Gen>.

**<Gen>** Generator number

**<Boolean>** ON - Power at port <Pt> is permanently on
OFF - Power is only on for the partial measurements that require the port as a drive port.

*RST value OFF

**SCPI, Command Types** Device-specific, command or query

**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 a permanent external generator signal.

## SOURce<Ch>:POWer<Pt>[:LEVel][:IMMediate][:AMPlitude] <numeric_value>

Defines the power of the internal signal source (channel power). The setting is valid for all sweep types except power sweep.

Use `SOURce<Ch>:POWer<Pt>:STARt` and `SOURce<Ch>:POWer<Pt>:STOP` to define the sweep range for a power sweep.

| | |
|---|---|
| **<Ch>** | Channel number. |
| **<Pt>** | Test port number of the analyzer. This suffix is ignored because the selected channel 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<Pt>[:LEVel][:IMMediate]:SLOPe`). Moreover, it is possible to define a port-specific power offset (`SOURce<Ch>:POWer<Pt>[:LEVel][:IMMediate]:OFFSet`). |
| **<numeric_value>** | Internal source power. |
| Range [def. unit] | –40 dBm to +10 dBm. The exact range depends on the analyzer model; refer to the data sheet [dBm]. UP and DOWN increment/decrement the source power in 0.1-dB steps. |
| *RST value | 0 dBm |
| **SCPI, Command Types** | Confirmed, command or query |

**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.

```
SOUR:POW2:SLOP 10
```

Set the power slope factor for the wave transmitted at port 2 to 10 dB/GHz. The internal source power and the waves at the other test ports are not affected.

## SOURce<Ch>:POWer<Pt>[:LEVel][:IMMediate]:OFFSet <numeric_value>, ONLY | CPADd

Defines a port-specific source power or a power offset relative to the channel power (`SOURce<Ch>:POWer<Pt>[:LEVel][:IMMediate][:AMPlitude]`). An additional *Cal Power Offset* can be defined via `SOURce<Ch>:POWer<Pt>:CORRection:LEVel:OFFSet`.

| | |
|---|---|
| **<Ch>** | Channel number. |
| **<Pt>** | Test port number of the analyzer. |
| **<numeric_value>** | Port-specific power offset. |
| Range [def. unit] | –300 dB to +300 dB [dB]. UP and DOWN increment/decrement the phase in 1-dB steps. |
| *RST value | 0 dB |
| **Offset parameters** | ONLY – only the port-specific power is used; the channel power is ignored.<br>CPADd – the port-specific power is added as an offset to the channel power. |
| *RST value | CPADd |
| **SCPI, Command Types** | Confirmed (with additional device-specific character data parameters), command or query |

**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.

## SOURce<Ch>:POWer<Pt>[:LEVel][:IMMediate]:SLOPe <numeric_value>

Defines a linear factor to modify the internal source power at port <Pt> as a function of the stimulus frequency. The value can be set for frequency sweeps only.

**<Ch>** Channel number.

**<Pt>** Test port number of the analyzer.

**<numeric_value>** Port-specific slope factor.

Range [def. unit] –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 [dBm]. UP and DOWN increment/decrement the source power in 0.1-dB/GHz steps.

*RST value 0 dB/GHz

**SCPI, Command Types** Device-specific, command or query

**Example:**

```
See SOURce<Ch>:POWer<Pt>[:LEVel][:IMMediate][:AMPlitude]
```

## SOURce<Ch>:POWer<Pt>:PERManent[:STATe] <Boolean>

Defines whether the source power is permanently on.

**<Ch>** Channel number.

**<Pt>** Test port number of the analyzer.

**<Boolean>** ON - Power at port <Pt> is permanently on
OFF - Power is only on for the partial measurements that require the port as a drive port.

*RST value OFF

**SCPI, Command Types** Device-specific, command or query

**Example:**

```
SOUR:POW2:STAT ON
```

Switch the RF power at port no. 2 on.

```
SOUR:POW2:PERM ON
```

Activate a permanent signal source.

## SOURce<Ch>:POWer<Pt>:STARt <numeric_value>

Defines the start power for a power sweep which is equal to the left edge of a Cartesian diagram. A power sweep must be active (`[SENSe<Ch>:]SWEep:TYPE POWer`) before this command can be used.

**<Ch>** Channel number.

**<Pt>** 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.

**<numeric_value>** Start power of the sweep.

Range [def. unit] –40 dBm to +9.99 dBm. The exact range depends on the analyzer model; refer to the data sheet [dBm]. UP and DOWN increment/decrement the source power in 1-dB steps.

*RST value –25 dBm

**SCPI, Command Types** Confirmed, command or query

**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.

***Note:*** *If the start power entered is greater than the current stop power (`SOURce<Ch>:POWer<Pt>:STOP`), the stop power is set to the start power plus the minimum power span of 0.01 dB.*

## SOURce<Ch>:POWer<Pt>:STATe <Boolean>

Turns the RF source power at a specified test port on or off.

**<Ch>** Channel number.

**<Pt>** Test port number of the analyzer.

**<Boolean>** ON | OFF - Turns the internal source power at the specified test port no. <Pt> on or off.

*RST value ON

**SCPI, Command Types** Device-specific, command or query

**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.

### SOURce<Ch>:POWer<Pt>:STOP <numeric_value>

Defines the stop power for a power sweep which is equal to the right edge of a Cartesian diagram. A power sweep must be active (`[SENSe<Ch>:]SWEep:TYPE POWer`) before this command can be used.

| | |
|---|---|
| **<Ch>** | Channel number. |
| **<Pt>** | 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. |
| **<numeric_value>** | Stop power of the sweep. |
| Range [def. unit] | –39.99 dBm to +10 dBm. The exact range depends on the analyzer model; refer to the data sheet [dBm]. UP and DOWN increment/decrement the source power in 1-dB steps. |
| *RST value | 0 dBm |
| **SCPI, Command Types** | Confirmed, command or query |
| **Example:** | `SWE:TYPE POW`<br>Activate a power sweep.<br>`SOUR:POW:STAR -6; STOP 10`<br>Select a power sweep range between –6 dBm and +10 dBm. |

*If the stop power entered is smaller than the current start power (`SOURce<Ch>:POWer<Pt>:STARt`), the start power is set to the stop power minus the minimum power span of 0.01 dB.*

## SOURce<Ch>:POWer:CORRection...

This subsystem controls the source power calibration.

***Note:*** *The SOURce<Ch>:POWer... subsystem comprises port-specific and general settings. Port-specific settings are valid for the port specified by the numeric suffix `<Pt>` (...`:POWER<Pt>:`...). General settings are valid for all test ports of the analyzer; the port suffix is ignored. Refer to the description of the individual commands for more information.*

### SOURce<Ch>:POWer<Pt>:CORRection[:ACQuire] PORT | GENerator | A1 | A2 | A3 | A4 | ESRC1 | ESRC2,<port_number>

Selects the source for the source power calibration, starts and applies the source power calibration.

*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.*

| | |
|---|---|
| **<Ch>** | Calibrated channel number |
| **<pt>** | Calibrated port number. This suffix is ignored because the port number is specified in the parameter list. |
| **PORT** | Analyzer port is the source of the calibrated wave. |
| **GENerator** | External generator is the source of the calibrated wave. The port and generator numbers are specified by means of the |

<port_number> 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.

**<port_number>** Analyzer port number of generator number, if PORT or GENerator is the first parameter.

Range [def. unit] 1 to port number of the analyzer/number of external generators [–]

*RST values – (for all parameters)

**SCPI, Command Types** Device-specific, no query

**Example:**

```
*RST; SOUR:POW:CORR:COLL:VER 0
```

Disable the verification sweep to speed up the source power calibration procedure.

```
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.

```
SOUR:POW:CORR:ACQ PORT,1
```

Perform a source power calibration using port 1 as a source port.

```
SOUR:POW:CORR?
```

Check whether the calibration is applied (the response is *1)*. Change the test setup, connect an 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)*.

## SOURce<Ch>:POWer<Pt>:CORRection:COLLect[:ACQuire] ASENsor | BSENsor

Initiates a source power calibration for the source port <pt> 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.*

**<Ch>** Calibrated channel number

**<Pt>** Calibrated port number.

**ASENsor | BSENsor** External power meter Pmtr 1
External power meter Pmtr 2

*RST value ASENsor

**SCPI, Command Types** Device-specific, no query

**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.

## SOURce<Ch>:POWer<Pt>:CORRection[:ACQuire]:VERification:RESult?

Returns the result of the last verification sweep. This command requires an enabled verification sweep (`SOURce<Ch>:POWer<Pt>:CORRection:COLLect[:ACQuire]:VERification[:STATe] ON`, see example below).

**<Ch>** Calibrated channel number. This parameter is ignored; the result is based on the last verification sweep acquired.

**<Pt>** Calibrated port number. This parameter is ignored; the result is based on the last verification sweep acquired.

**Response:** Result of the sweep in the format <Boolean>, <Max. Offset> where:

**<Boolean>** Calibration passed, i.e. the cal. power offset is below the tolerance (1), or failed (0).

**<Max. Offset>** Maximum power offset between the measured power at the reference plane and the *Cal Power* during the verification sweep in dB.

*RST value –
If no verification sweep result is available, the command returns 0.

**SCPI, Command Types** Device-specific, command or query

**Example:**

```
*RST; SOUR:POW:CORR:COLL:VER ON
```

Enable the verification 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.

```
SOUR:POW:CORR:ACQ PORT,1
```

Perform a source power calibration using port 1 as a source port.

```
SOUR:POW:CORR:COLL:VER:RES?
```

Query the result of the verification sweep.

## SOURce<Ch>:POWer<Pt>:CORRection:COLLect[:ACQuire]:VERification[:STATe] <Boolean>

Enables or disables a verification sweep that the analyzer performs after the source power calibration. The command is valid for all channels, ports and external generators.

**<Ch>** Calibrated channel number. This parameter is ignored; the verification is either enabled or disabled for all channels.

**<Pt>** Calibrated port number. This parameter is ignored; the verification is either enabled or disabled for all sources.

**<Boolean>** Enables (ON) or disables (OFF) the verification sweep.

*RST value ON

**SCPI, Command Types** Device-specific, command or query

**Example:**

```
See SOURce<Ch>:POWer<Pt>:CORRection[:ACQuire],
SOURce<Ch>:POWer<Pt>:CORRection[:ACQuire]:VERification:RESult?
```

## SOURce<Ch>:POWer<Pt>:CORRection:COLLect:AVERage[:COUNt] <readings>

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<Ch>:POWer<Pt>:CORRection:NREadings`.

**<Ch>** Calibrated channel number. This parameter is ignored; the limit applies to all channels.

**<Pt>** Calibrated port number. This parameter is ignored; the limit applies to all sources.

**<readings>** Number of readings

Range [def. unit] 1 to 100 [–]

*RST value 1

**SCPI, Command Types** Device-specific, command or query

**Example:**

```
See SOURce<Ch>:POWer<Pt>:CORRection[:ACQuire] (for equivalent
command SOURce<Ch>:POWer<Pt>:CORRection:NREadings).
```

## SOURce<Ch>:POWer<Pt>: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.

**<Ch>** Calibrated channel number. This parameter is ignored; the tolerance value is valid for all sources.

**<Pt>** Calibrated port number. This parameter is ignored; the tolerance

value is valid for all sources.

**<tolerance>** Tolerance value
Range [def. unit] 0.001 dB to 1000 dB [dB]
*RST value 1 dB

**SCPI, Command Types** Device-specific, command or query

**Example:**

```
See SOURce<Ch>:POWer<Pt>:CORRection[:ACQuire].
```

## SOURce<Ch>:POWer<Pt>: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 (SOURce<Ch>:POWer<Pt>:CORRection:COLLect:RRECeiver).*

**<Ch>** Calibrated channel number. This parameter is ignored; the verification is either enabled or disabled for all channels.

**<Pt>** Calibrated port number. This parameter is ignored; the verification is either enabled or disabled for all sources.

**<Boolean>** Enables (ON) or disables (OFF) 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 value ON

**SCPI, Command Types** Device-specific, command or query

**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<Ch>:POWer<Pt>:CORRection:COLLect:RRECeiver <Boolean>

Enables or disables calibration of the reference receiver together with the source power calibration.

*It is not possible to disable flatness calibration (SOURce<Ch>:POWer<Pt>:CORRection:COLLect:FLATness) and the reference receiver calibration simultaneously.*

| | |
|---|---|
| **<Ch>** | Calibrated channel number. This parameter is ignored; the verification is either enabled or disabled for all channels. |
| **<Pt>** | Calibrated port number. This parameter is ignored; the verification is either enabled or disabled for all sources. |
| **<Boolean>**<br>*RST value | Enables (ON) or disables (OFF) reference receiver calibration.<br>ON |
| **SCPI, Command Types** | Device-specific, command or query |
| **Example:** | `See SOURce<Ch>:POWer<Pt>:CORRection:COLLect:FLATness` |

## SOURce<Ch>:POWer<Pt>:CORRection:DATA '<src_string>', <block> | <numeric_value>...

Reads or writes 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.

| | |
|---|---|
| **<Ch>** | Calibrated channel number |
| **<Pt>** | Calibrated port number |
| **'<src_string>'** | Identifier for the source of the calibrated wave:<br>• 'A<n>' or 'PORT<n>' denote correction data for the analyzer port no. <n>.<br>• 'ESRC<n>' or 'GEN<n>' denote correction data for the external generator no. <n>. |
| **<block> | <numeric_value>...** | Power correction values in block data format or as a comma-separated list of real numbers. The query returns a list of numbers. |
| Range [def. unit] | All numbers are interpreted as dB-values. |
| *RST value | – |
| **SCPI, Command Types** | Device-specific, command or query |
| **Example:** | `*RST; :SWE:POIN 10`<br>Reset the instrument and reduce the number of sweep points to 10.<br>`SOUR:POW:CORR:ACQ PORT,1`<br>Perform a source power calibration using port 1 as a source port.<br>`SOUR:POW:CORR:DATA? 'A1'`<br>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 ten (new) numbers.

## SOURce<Ch>:POWer<Pt>: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.

| | |
|---|---|
| **<Ch>** | Calibrated channel number |
| **<Pt>** | Calibrated port number. This suffix is ignored; the generator is selected via <Gen>. |
| **<Gen>** | Generator number |
| **<offset>** | Gain or attenuation value |
| Range [def. unit] | –300 dB to +300 dB [dB] |
| *RST value | 0 dB |
| **SCPI, Command Types** | Device-specific, command or query |

**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.

## SOURce<Ch>:POWer<Pt>: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<Pt>:CORRection[:ACQuire]`). To enable or disable a source power calibration for an analyzer port use `SOURce<Ch>:POWer<Pt>:CORRection[:STATe]`.

| | |
|---|---|
| **<Ch>** | Calibrated channel number |
| **<Pt>** | Calibrated external generator. This suffix is ignored; the generator is selected via <Gen>. |
| **<Gen>** | Generator number |
| **<Boolean>** | Enables (ON) or disables (OFF) the source power calibration for port number <Pt>. |
| *RST value | OFF [–] |
| **SCPI, Command Types** | Device-specific, command or query |
| **Example:** | `See SOURce<Ch>:POWer<Pt>:CORRection[:ACQuire].` |

## SOURce<Ch>:POWer<Pt>:CORRection:HARMonic[:ACQuire]

Starts the source calibration (1st power calibration step for harmonic measurements), stores and applies the calibration data. The external power meter used is selected via `SOURce<Ch>:POWer<Pt>:CORRection:PMETer:ID`.

| | |
|---|---|
| **<Ch>** | Calibrated channel number |
| **<pt>** | Port number This suffix is ignored; the analyzer calibrates the port that is selected as a source port for the harmonic measurement (`[SENSe<Ch>:]FREQuency:CONVersion:HARMonic:SPORt`). |
| **SCPI, Command Types** | Device-specific, no query |
| **Example:** | `*RST; FREQ:STAR 0.5E+9; STOP 1E+9` |

Reset the analyzer (activating a frequency sweep) and set the sweep range between 1 GHz and 1.1 GHz.

```
FREQ:CONV:HARM:ORD 4; SPOR 1; RPOR 2
```

Configure a 4th harmonic measurement with source port 1 and receive port 2. The measurement is still not enabled.

```
FREQ:CONV HARM
```

Enable the configured harmonic measurement.

```
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:HARM
```

Start source power calibration at the previously selected source port 1.

```
CORR:POW:HARM:ACQ
```

Start receiver power calibration at the previously selected receive port 2.

```
CALC:PAR:MEAS? "TRC1"
```

Query the measured quantity. The analyzer measures the wave quantity b2, the response is `'B2D1SAM'`.

## SOURce<Ch>:POWer<Pt>: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.

**<Ch>** Calibrated channel number

**<Pt>** Calibrated port number

**<offset>** Gain or attenuation value

Range [def. unit] –300 dB to +300 dB [dB]

*RST value 0 dB

**SCPI, Command Types** Device-specific, command or query

**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.

## SOURce<Ch>:POWer<Pt>:CORRection:MIXer:LO[:ACQuire]

Starts the LO source calibration (3rd power calibration step for mixer measurements), stores and applies the calibration data. The external power meter is selected via `SOURce<Ch>:POWer<Pt>:CORRection:PMETer:ID`.

*The LO source calibration is independent of the RF source and IF receiver calibration; see description of the mixer calibration procedure.*

**<Ch>** Calibrated channel number

**<pt>** Calibrated port number. This suffix is ignored; the port is selected via `[SENSe<Ch>:]FREQuency:CONVersion:MIXer:LOINternal` or `[SENSe<Ch>:]FREQuency:CONVersion:MIXer:LOEXternal`.

**SCPI, Command Types** Device-specific, no query

**Example:**

```
See SOURce<Ch>:POWer<Pt>:CORRection:MIXer:RF[:ACQuire].
```

## SOURce<Ch>:POWer<Pt>:CORRection:MIXer:RF[:ACQuire]

Starts the RF source calibration (1st power calibration step for mixer measurements), stores and applies the calibration data. The external power meter used is selected via `SOURce<Ch>:POWer<Pt>:CORRection:PMETer:ID`.

*The command initiates two calibration steps, the first at the RF frequency range, the second at the IF frequency (for a subsequent receiver power calibration, [SENSe<Ch>:]`CORRection:POWer<port_no>`:MIXer:IF:ACQuire). See description of the mixer calibration procedure.*

**<Ch>** Calibrated channel number

**<pt>** Calibrated port number. This suffix is ignored because the RF signal is always assigned to port 1.

**SCPI, Command Types** Device-specific, no query

**Example:**

```
FREQ:CONV MIX
...
```

Select a scalar mixer measurement; set frequencies, see e.g. example for `[SENSe<Ch>:]FREQuency:CONVersion:MIXer:TFRequency`.

```
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:MIX:RF
```

Start the RF source calibration. Change the test setup for the IF receiver calibration (see mixer calibration procedure).

```
CORR:POWer:MIX:IF:ACQuire
```

Start the IF receiver calibration.

```
FREQ:CONV:MIX:LOIN 3
```

Select port no. 3 as a signal source for the LO signal. Adjust the test setup for the LO source calibration.

```
SOUR:POW:CORR:MIX:LO
```

Start the LO source calibration.

## SOURce<Ch>:POWer<Pt>:CORRection:NREadings <readings>

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<Ch>:POWer<Pt>:CORRection:COLLect:AVERage[:COUNt]`.

**<Ch>** Calibrated channel number. This parameter is ignored; the limit applies to all sources.

| | |
|---|---|
| **<Pt>** | Calibrated port number. This parameter is ignored; the limit applies to all sources. |
| **<readings>** | Number of readings |
| Range [def. unit] | 1 to 100 [–] |
| *RST value | 1 |
| **SCPI, Command Types** | Device-specific, command or query |
| **Example:** | `See SOURce<Ch>:POWer<Pt>:CORRection[:ACQuire].` |

## SOURce<Ch>:POWer<Pt>:CORRection:OSOurces[:STATe] <Boolean>

Switches off all other sources during the calibration sweep for channel <Ch>.

| | |
|---|---|
| **<Ch>** | Calibrated channel number |
| **<Pt>** | Calibrated port number. This parameter is ignored; the setting is valid for all sources. |
| **<Boolean>** | ON – Other sources not necessarily switched off<br>OFF – Other sources switched off during the calibration sweep |
| *RST value | ON [–] |
| **SCPI, Command Types** | Device-specific, command or query |
| **Example:** | `See SOURce<Ch>:POWer<Pt>:CORRection[:ACQuire].` |

## SOURce<Ch>:POWer<Pt>:CORRection:PMETer:ID <pmeter_no>

Selects an external power meter for the source power calibration.

*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.*

| | |
|---|---|
| **<Ch>** | Calibrated channel number |
| **<Pt>** | Calibrated port number. This parameter is ignored; the power meter selection is valid for all sources. |
| **<pmeter_no>** | Number of external power meter. The parameters UP, DOWN, MIN, MAX are not available for this command. |
| Range [def. unit] | 1 to number of external generators [–] |
| *RST value | The power meter selection is not changed by a reset of the analyzer. |
| **SCPI, Command Types** | Device-specific, command or query. |
| **Example:** | `See SOURce<Ch>:POWer<Pt>:CORRection:COLLect[:ACQuire] and SOURce<Ch>:POWer<Pt>:CORRection:MIXer:RF[:ACQuire].` |

### SOURce<Ch>:POWer<Pt>:CORRection:STATe <Boolean>

Enables or disables the source power calibration for channel <Ch> and for port number <Pt>. The command is disabled unless a source power calibration for the analyzer port has been performed (`SOURce<Ch>:POWer<Pt>:CORRection[:ACQuire]`). To enable or disable a source power calibration for an external generator use `SOURce<Ch>:POWer<Pt>:CORRection:GENerator[:STATe]`.

| | |
|---|---|
| **<Ch>** | Calibrated channel number |
| **<Pt>** | Calibrated port number |
| **<Boolean>** | Enables (ON) or disables (OFF) the source power calibration for port number <Pt>. |
| *RST value | OFF [–] |
| **SCPI, Command Types** | Device-specific, command or query |
| **Example:** | `See SOURce<Ch>:POWer<Pt>:CORRection[:ACQuire].` |

## SOURce<Ch>:TDIF...

This subsystem controls the True Differential Mode (with option R&S ZVA-K6).

### SOURce<Ch>:TDIF:IMBalance:AMPLitude:LPORt <Log Port>

Selects a logical (balanced) port for the amplitude imbalance sweep in true differential mode. A balanced port configuration must be defined and an amplitude imbalance sweep must be active to use this command; see program example.

| | |
|---|---|
| **<Ch>** | Channel number. |
| **<Log Port>** | Logical port number of the balanced port. The balanced port must be defined using `SOURce<Ch>:LPORt<log_port>` before it can be selected. |
| Range [def. unit] | Depending on the number of balanced ports defined. [–] |
| **SCPI, Command Types** | Device-specific, command or query |
| **Example:** | `*RST; SOUR:LPOR1 1,3` |
| | Reset the analyzer and combine the physical ports 1 and 3 to the logical port 1. |
| | `SWE:TYPE IAMP; :SOUR:TDIF?` |
| | Select an amplitude imbalance sweep. This sweep type automatically activates the true differential mode (the query returns 1). |
| | `CALC:TDIF:IMB:COMP ON` |
| | Use imbalance-compensated wave quantities to calculate S-parameters, ratios and derived quantities. |
| | `SOUR:TDIF:IMB:AMPL:LPOR 1; STAR -20; STOP 0` |
| | Configure the amplitude imbalance sweep, selecting the logical port no. 1 and a swept power range between -20 dBm and 0 dBm at the physical port no. 3 (i.e. at the physical port with the highest port number that belongs to the selected logical port). |
| | `CALC:PAR:SDEF 'Trc1', 'SDS12'` |

Create a new trace named 'Trc1' to measure the mixed mode S-parameter $S_{ds12}$.

```
DISP:TRAC:FEED 'Trc1'
```

Display the result

## SOURce<Ch>:TDIF:IMBalance:AMPLitude:STARt <Start Power>

Defines the start power for an amplitude imbalance sweep, which is equal to the left edge of a Cartesian diagram. A balanced port configuration must be defined and an amplitude imbalance sweep must be active to use this command; see program example.

| | |
|---|---|
| **<Ch>** | Channel number. |
| **<Start Power>** | Start power of the sweep. |
| Range [def. unit] | –40 dBm to +9.99 dBm. The exact range depends on the analyzer model; refer to the data sheet [dBm]. UP and DOWN increment/decrement the source power in 0.01-dB steps. The start power must be smaller than the stop power (`SOURce<Ch>:TDIF:IMBalance:AMPLitude:STOP`). |
| *RST value | –10 dBm |
| **SCPI, Command Types** | Device-specific, command or query |
| **Example:** | `See SOURce<Ch>:TDIF:IMBalance:AMPLitude:LPORt` |

## SOURce<Ch>:TDIF:IMBalance:AMPLitude:STOP <Stop Power>

Defines the stop power for an amplitude imbalance sweep, which is equal to the right edge of a Cartesian diagram. A balanced port configuration must be defined and an amplitude imbalance sweep must be active to use this command; see program example.

| | |
|---|---|
| **<Ch>** | Channel number. |
| **<Stop Power>** | Stop power of the sweep. |
| Range [def. unit] | –39.99 dBm to +10 dBm. The exact range depends on the analyzer model; refer to the data sheet [dBm]. UP and DOWN increment/decrement the source power in 0.01-dB steps. The stop power must be larger than the start power (`SOURce<Ch>:TDIF:IMBalance:AMPLitude:STARt`). |
| *RST value | +10 dBm |
| **SCPI, Command Types** | Device-specific, command or query |
| **Example:** | `See SOURce<Ch>:TDIF:IMBalance:AMPLitude:LPORt` |

## SOURce<Ch>:TDIF:IMBalance:PHASe:LPORt <Log Port>

Selects a logical (balanced) port for the phase imbalance sweep in true differential mode. A balanced port configuration must be defined and a phase imbalance sweep must be active to use this command; see program example.

**<Ch>** Channel number.

**<Log Port>** Logical port number of the balanced port. The balanced port must be defined using `SOURce<Ch>:LPORt<log_port>` before it can be selected.

Range [def. unit] Depending on the number of balanced ports defined. [–]

**SCPI, Command Types** Device-specific, command or query

**Example:**

```
*RST; SOUR:LPOR1 1,3
```

Reset the analyzer and combine the physical ports 1 and 3 to the logical port 1.

```
SWE:TYPE IPH; :SOUR:TDIF?
```

Select a phase imbalance sweep. This sweep type automatically activates the true differential mode (the query returns 1).

```
SOUR:TDIF:IMB:PHAS:LPOR 1; STAR -20; STOP 0
```

Configure the phase imbalance sweep, selecting the logical port no. 1 and a swept phase range between -20 deg and 0 deg at the physical port no. 3 (i.e. at the physical port with the highest port number that belongs to the selected logical port).

```
CALC:PAR:SDEF 'Trc1', 'SDS12'
```

Create a new trace named 'Trc1' to measure the mixed mode S-parameter $S_{ds12}$.

```
DISP:TRAC:FEED 'Trc1'
```

Display the result

## SOURce<Ch>:TDIF:IMBalance:PHASe:STARt <Start Phase>

Defines the start phase for a phase imbalance sweep, which is equal to the left edge of a Cartesian diagram. A balanced port configuration must be defined and a phase imbalance sweep must be active to use this command; see program example.

**<Ch>** Channel number.

**<Start Phase>** Start phase of the sweep.

Range [def. unit] –180 deg to +179.99 deg [deg]. UP and DOWN increment/decrement the phase in 1-deg / 0.1-deg / 0.01-deg steps. The start phase must be smaller than the stop phase (`SOURce<Ch>:TDIF:IMBalance:PHASe:STOP`).

*RST value –180 deg

**SCPI, Command Types** Device-specific, command or query

**Example:**

```
See SOURce<Ch>:TDIF:IMBalance:PHASe:LPORt
```

## SOURce<Ch>:TDIF:IMBalance:PHASe:STOP <Stop Phase>

Defines the stop phase for a phase imbalance sweep, which is equal to the right edge of a Cartesian diagram. A balanced port configuration must be defined and a phase imbalance sweep must be active to use this command; see program example.

| | |
|---|---|
| **<Ch>** | Channel number. |
| **<Stop Phase>** | Stop phase of the sweep. |
| Range [def. unit] | –179.99 deg to +180 deg [deg]. UP and DOWN increment/decrement the phase in 1-deg / 0.1-deg / 0.01-deg steps. The stop phase must be larger than the start phase (`SOURce<Ch>:TDIF:IMBalance:PHASe:STARt`). |
| *RST value | +180 deg |
| **SCPI, Command Types** | Device-specific, command or query |
| **Example:** | `See SOURce<Ch>:TDIF:IMBalance:PHASe:LPORt` |

## SOURce<Ch>:TDIF[:STATe] <Boolean>

Switches the true differential mode on or off. This command is available only if a suitable balanced port configuration is active.

**Balanced port configurations for true differential mode**

The true differential mode requires at least one balanced port. The two physical ports of the balanced port must be supplied by different power sources. This means that the analyzer port combinations 1/2, 3/4 are forbidden.

The number of possible port configurations increases with the number of analyzer ports:

- For 2-port analyzers, no suitable balanced port configuration is possible.
- For 3-port analyzers, either ports no. 1 and 3 or ports no. 2 and 3 can be combined to form a single balanced port. The third port is single-ended (unbalanced).
- For 4-port analyzers, the port combinations 1/3, 1/4, 2/3, 2/4 are allowed. The analyzer may contain either one or two balanced ports.

| | |
|---|---|
| **<Ch>** | Channel number. |
| **<Boolean>** | ON - True differential mode active<br>OFF - True differential mode inactive, the virtual differential mode provides balanced results |
| *RST value | OFF |
| **SCPI, Command Types** | Device-specific, command or query |
| **Example:** | `*RST; SOUR:LPOR1 1,3` |
| | Reset the analyzer and combine the physical ports 1 and 3 to the logical port 1. Defining a balanced port configuration deletes the default trace. |
| | `SOUR:TDIF ON` |
| | Switch true differential mode on. |
| | `CALC:PAR:SDEF 'Trc1', 'BD1D2S'` |

Create a new trace to measure the differential received wave quantity $b_{d1}$, using the single-ended analyzer port no. 2 as a source port.

### SOURce<Ch>:TDIF:WAVes SENDed | DCMode

Qualifies whether the source power (`SOURce<Ch>:POWer<Pt>[:LEVel][:IMMediate][:AMPlitude]`) is equal to the power of each single-ended wave or to the balanced waves in true differential mode.

| | |
|---|---|
| **<Ch>** | Channel number. |
| **SENDed \| DCMode** | Single-ended: Same Differential and Common Mode Voltages as in Single-Ended Mode.<br>Differential and common mode: Apply to Differential (Zd = 2Z0) and Common Mode (Zd = Z0/2) Waves. |
| *RST value | – (a reset leaves the setting unchanged) |
| **SCPI, Command Types** | Device-specific, command or query |
| **Example:** | `*RST; SOUR:LPOR1 1,3` |
| | Reset the analyzer and combine the physical ports 1 and 3 to the logical port 1. Defining a balanced port configuration deletes the default trace. |
| | `SOUR:TDIF ON` |
| | Switch true differential mode on. |
| | `SOUR:TDIF:WAV SEND` |
| | Select single-ended source power mode. |

## STATus...

This subsystem controls the Status Reporting System. Note that *RST does not influence the status registers.

### 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 *Reset Values of the Status Reporting System*.

| | |
|---|---|
| ***RST value** | – |
| **SCPI, Command Types** | Confirmed, no query |

**Example:**
```
STAT:PRES
```
Preset the status registers.

## STATus:QUEStionable:CONDition?

Returns the contents of the CONDition part of the QUEStionable register. Reading the CONDition registers is nondestructive.

***RST value** –

**SCPI, Command Types** Confirmed, query only

**Example:**
```
STAT:QUES:COND?
```
Query the CONDition part of the QUEStionable register to check for questionable instrument states.

## STATus:QUEStionable:ENABle

Sets the enable mask which allows true conditions in the EVENt part of the QUEStionable register 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 (bit 3 of the STatus Byte).

**<NRf>** 0 to 65535 (decimal representation)

*RST value – (see also *Reset Values of the Status Reporting System)*

**SCPI, Command Types** Confirmed, command or query

**Example:**
```
STAT:QUES:ENABle 1536
```
Set bits no. 9 and 10 of the QUEStionable:ENABle register

## STATus:QUEStionable[:EVENt]?

Returns the contents of the EVENt part of the QUEStionable register. Reading an EVENt register clears it.

***RST value** –

**SCPI, Command Types** Confirmed, query only

**Example:**
```
STAT:OPER?
```
Query the EVENt part of the OPERation register to check whether an event has occurred since the last reading.

## STATus:QUEStionable:NTRansition

Sets the negative transition filter. Setting a bit causes a 1 to 0 transition in the corresponding bit of the associated condition register to cause a 1 to be written in the associated bit of the corresponding event register.

| | |
|---|---|
| **<NRf>** | 0 to 65535 (decimal representation) |
| *RST value | – (see also *Reset Values of the Status Reporting System)* |
| **SCPI, Command Types** | Confirmed, command or query |
| **Example:** | `STAT:QUES:NTR 1536` |
| | Set bits no. 9 and 10 of the QUEStionable:NTRansition register |

## STATus:QUEStionable:PTRansition

Sets the positive transition filter. Setting a bit causes a 0 to 1 transition in the corresponding bit of the associated condition register to cause a 1 to be written in the associated bit of the corresponding event register.

| | |
|---|---|
| **<NRf>** | 0 to 65535 (decimal representation) |
| *RST value | – (see also *Reset Values of the Status Reporting System)* |
| **SCPI, Command Types** | Confirmed, command or query |
| **Example:** | `STAT:QUES:PTR 1536` |
| | Set bits no. 9 and 10 of the QUEStionable:PTRansition register |

## STATus:QUEStionable:INTegrity:CONDition?

Returns the contents of the CONDition part of the QUEStionable:INTegrity register. Reading the CONDition registers is nondestructive.

| | |
|---|---|
| ***RST value** | – |
| **SCPI, Command Types** | Confirmed, query only |
| **Example:** | `STAT:QUES:INT:COND?` |
| | Query the CONDition part of the QUEStionable:INTegrity register to detect current hardware failures. |

## STATus:QUEStionable:INTegrity:ENABle

Sets the enable mask which allows true conditions in the EVENt part of the QUEStionable:INTegrity register 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 (bit 9 of the QUEStionable register).

| | |
|---|---|
| **<NRf>** | 0 to 65535 (decimal representation) |
| *RST value | – (see also *Reset Values* of the status reporting system) |
| **SCPI, Command Types** | Confirmed, command or query |
| **Example:** | `STAT:QUES:INT:ENAB 4` |
| | Set bit no. 2 of the QUEStionable:INTegrity:ENABle register |

## STATus:QUEStionable:INTegrity[:EVENt]?

Returns the contents of the EVENt part of the QUEStionable:INTegrity register. Reading an EVENt register clears it.

| | |
|---|---|
| ***RST value** | – |
| **SCPI, Command Types** | Confirmed, query only |
| **Example:** | `STAT:QUES:INT?` |
| | Query the EVENt part of the QUEStionable:INTegrity register to check whether an event has occurred since the last reading. |

## STATus:QUEStionable:INTegrity:HARDware:CONDition?

Returns the contents of the CONDition part of the QUEStionable:INTegrity:HARDware register. Reading the CONDition registers is nondestructive.

| | |
|---|---|
| ***RST value** | – |
| **SCPI, Command Types** | Confirmed, query only |
| **Example:** | `STAT:QUES:INT:HARD:COND?` |
| | Query the CONDition part of the QUEStionable:INTegrity:HARDware register to detect current hardware failures. |

## STATus:QUEStionable:INTegrity:HARDware:ENABle

Sets the enable mask which allows true conditions in the EVENt part of the QUEStionable:INTegrity:HARDware register 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 (bit 2 of the QUEStionable:INTegrity register).

| | |
|---|---|
| **<NRf>** | 0 to 65535 (decimal representation) |
| *RST value | – (see also *Reset Values* of the status reporting system) |
| **SCPI, Command Types** | Confirmed, command or query |
| **Example:** | `STAT:QUES:INT:HARD:ENAB 394` |

Set bits no. 1, 3, 7, and 8 of the
QUEStionable:INTegrity:HARDware:ENABle register

## STATus:QUEStionable:INTegrity:HARDware[:EVENt]?

Returns the contents of the EVENt part of the QUEStionable:INTegrity:HARDware register. Reading an EVENt register clears it.

***RST value** –

**SCPI, Command Types** Confirmed, query only

**Example:**

```
STAT:QUES:INT:HARD?
```

Query the EVENt part of the QUEStionable:INTegrity:HARDware register to check whether an event has occurred since the last reading.

## STATus:QUEStionable:INTegrity:HARDware:NTRansition

Sets the negative transition filter. Setting a bit causes a 1 to 0 transition in the corresponding bit of the associated condition register to cause a 1 to be written in the associated bit of the corresponding event register.

**<NRf>** 0 to 65535 (decimal representation)
*RST value – (see also *Reset Values* of the status reporting system)

**SCPI, Command Types** Confirmed, command or query

**Example:**

```
STAT:QUES:INT:HARD:NTR 394
```

Set bits no. 1, 3, 7, and 8 of the
QUEStionable:INTegrity:HARDware:NTRansition register

## STATus:QUEStionable:INTegrity:HARDware:PTRansition

Sets the positive transition filter. Setting a bit causes a 0 to 1 transition in the corresponding bit of the associated condition register to cause a 1 to be written in the associated bit of the corresponding event register.

**<NRf>** 0 to 65535 (decimal representation)
*RST value – (see also *Reset Values* of the status reporting system)

**SCPI, Command Types** Confirmed, command or query

**Example:**

```
STAT:QUES:INT:HARD:PTR 394
```

Set bits no. 1, 3, 7, and 8 of the

QUEStionable:INTegrity:HARDware:PTRansition register

## STATus:QUEStionable:INTegrity:NTRansition

Sets the negative transition filter. Setting a bit causes a 1 to 0 transition in the corresponding bit of the associated condition register to cause a 1 to be written in the associated bit of the corresponding event register.

| | |
|---|---|
| **<NRf>** | 0 to 65535 (decimal representation) |
| *RST value | – (see also *Reset Values* of the status reporting system) |
| **SCPI, Command Types** | Confirmed, command or query |
| **Example:** | `STAT:QUES:INT:NTR 4` |
| | Set bit no. 2 of the QUEStionable:INTegrity:NTRansition register |

## STATus:QUEStionable:INTegrity:PTRansition

Sets the positive transition filter. Setting a bit causes a 0 to 1 transition in the corresponding bit of the associated condition register to cause a 1 to be written in the associated bit of the corresponding event register.

| | |
|---|---|
| **<NRf>** | 0 to 65535 (decimal representation) |
| *RST value | – (see also *Reset Values* of the status reporting system) |
| **SCPI, Command Types** | Confirmed, command or query |
| **Example:** | `STAT:QUES:INT:PTR 4` |
| | Set bit no. 2 of the QUEStionable:INTegrity:PTRansition register |

## STATus:QUEStionable:LIMit<1|2>:CONDition?

Returns the contents of the CONDition part of the QUEStionable:LIMit<1|2> register. Reading the CONDition registers is nondestructive.

| | |
|---|---|
| ***RST value** | – |
| **SCPI, Command Types** | Confirmed, query only |
| **Example:** | `STAT:QUES:LIMit:COND?` |
| | Query the CONDition part of the QUEStionable:LIMit1 register to retrieve the current status of the limit check. |

## STATus:QUEStionable:LIMit<1|2>:ENABle

Sets the enable mask which allows true conditions in the EVENt part of the QUEStionable:LIMit<1|2> register 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 (bit 10 of the QUEStionable register for the LIMit1 register, bit 0 of the LIMit1 register for the LIMit2 register).

| | |
|---|---|
| **<NRf>** | 0 to 65535 (decimal representation) |
| *RST value | – (see also *Reset Values* of the status reporting system) |
| **SCPI, Command Types** | Confirmed, command or query |
| **Example:** | `STAT:QUES:LIM2:ENAB 6` |
| | Set bits no. 1 and 2 of the QUEStionable:LIMit2:ENABle register |

## STATus:QUEStionable:LIMit<1|2>[:EVENt]?

Returns the contents of the EVENt part of the QUEStionable:LIMit<1|2> register. Reading an EVENt register clears it.

| | |
|---|---|
| ***RST value** | – |
| **SCPI, Command Types** | Confirmed, query only |
| **Example:** | `STAT:QUES:LIM1?` |
| | Query the EVENt part of the QUEStionable:LIMit1 register to check whether an event has occurred since the last reading. |

## STATus:QUEStionable:LIMit<1|2>:NTRansition

Sets the negative transition filter. Setting a bit causes a 1 to 0 transition in the corresponding bit of the associated condition register to cause a 1 to be written in the associated bit of the corresponding event register.

| | |
|---|---|
| **<NRf>** | 0 to 65535 (decimal representation) |
| *RST value | – (see also *Reset Values of the Status Reporting System)* |
| **SCPI, Command Types** | Confirmed, command or query |
| **Example:** | `STAT:QUES:LIM2:NTR 6` |
| | Set bits no. 1 and 2 of the QUEStionable:LIMit2:NTRansition register |

## STATus:QUEStionable:LIMit<1|2>:PTRansition

Sets the positive transition filter. Setting a bit causes a 0 to 1 transition in the corresponding bit of the associated condition register to cause a 1 to be written in the associated bit of the corresponding event register.

| | |
|---|---|
| **<NRf>** | 0 to 65535 (decimal representation) |
| *RST value | – (see also *Reset Values of the Status Reporting System)* |
| **SCPI, Command Types** | Confirmed, command or query |
| **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 Error Messages.

| | |
|---|---|
| ***RST value** | – |
| **SCPI, Command Types** | Confirmed, query only |
| **Example:** | `STAT:QUE?` |
| | Query the oldest entry in the error queue. 0,"No error" is returned if the error queue is empty. |

## SYSTem...

This subsystem collects the functions that are not related to instrument performance, such as function for general housekeeping and function related to global configurations.

### SYSTem:COMMunicate:AKAL:CONNection THRough | OPEN | SHORt | MATCh, <port_1>, <port_2>

Connects the selected calibration standard to one or two ports of the active calibration unit (see `SYSTem:COMMunicate:RDEVice:AKAL:ADDRess`).

| | |
|---|---|
| **THRough \| OPEN \| SHORt \| MATCh** | Connected one or two-port standard |
| **<port_1>** | Port number of the calibration unit, for one and two-port standards |
| **<port_2>** | Second port number of the calibration unit, for two-port standards (THRough) only, omitted for the one-port standards |
| *RST value | – |
| **SCPI, Command Types** | Device-specific, command or query |
| **Example:** | `SYST:COMM:AKAL:CONN THR, 1, 2` |
| | Connect a trough standard between ports 1 and 2 of the cal unit. |

## SYSTem:COMMunicate:AKAL:MMEMory[:STATe] <Boolean>

Shows or hides the memory of the active calibration unit (see `SYSTem:COMMunicate:RDEVice:AKAL:ADDRess`).

| | |
|---|---|
| **<Boolean>** | ON – memory is shown in a separate drive<br>OFF – memory is not shown |
| *RST value | OFF |
| **SCPI, Command Types** | Device-specific, command or query |
| **Example:** | `SYST:COMM:AKAL:MMEM ON` |
| | Show the memory of the active calibration unit. |

## SYSTem:COMMunicate:GPIB[:SELF]:ADDRess <address_no>

Sets the GPIB address of the analyzer.

| | |
|---|---|
| **<address_no>** | GPIB address (integer number) |
| Range [def. unit] | 0 to 30 [–]. |
| *RST value | The GPIB address is factory-preset. *RST has no effect on the value. |
| **SCPI, Command Types** | Confirmed, command or query |
| **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). |

## SYSTem:COMMunicate:GPIB[:SELF]:RTERminator LFEoi | EOI

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.

| | |
|---|---|
| **LFEoi** | LF character sequence with or without EOI recognized |
| **EOI** | EOI recognized only |
| *RST value | LFEoi. |
| **SCPI, Command Types** | Device-specific, command or query |

**Example:**

```
SYST:COMM:GPIB:RTER EOI
```

Set the terminator to EOI.

## SYSTem:COMMunicate:RDEVice:AKAL:ADDRess '<cal_unit_name>'

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.

**<cal_unit_name>** 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?`

*RST values '' (empty string; *RST has no effect on external devices).

**SCPI, Command Types** Device-specific, command or query

**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.

## SYSTem:COMMunicate:RDEVice:AKAL:ADDRess:ALL?

Queries the names (USB addresses) of all connected calibration units.

**Response** Names (USB addresses) of all connected calibration units (comma-separated list of string variables).

*RST values '' (empty string; *RST has no effect on external devices).

**SCPI, Command Types** Device-specific, query only

**Example:**

```
See SYSTem:COMMunicate:RDEVice:AKAL:ADDRess
```

## SYSTem:COMMunicate:RDEVice:GENerator<gen_no>:DEFine '<gen_name>', '<driver>', '<interface>', '<address>'[, <fast_sweep>, <10_MHz_Ref>]

Configures an external generator and adds it to the list of available generators.

**<gen_no>** Number of the configured generator. Generators must be numbered in ascending order, starting with 1. If a number is re-used, the previous generator configuration is overwritten.

| | |
|---|---|
| | Generators can be assigned several times so that the number of configured generators is practically unlimited. |
| **'<gen_name>'** | 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.<br>**Alternative:** The '<gen_name>' string may contain the generator driver file name with its complete path, e.g. 'C:\Program Files\Rohde&Schwarz\Network Analyzer\resources\extdev\sme02.gen'. |
| **'<interface>'** | Interface type (string parameter): 'GPIB0', 'VXI-11',... |
| **'<address>'** | Interface address (string parameter), depending on the interface type. |
| **<fast_sweep>** | Optional Boolean parameter, enables or disables the fast sweep mode. |
| **<10_MHz_Ref>** | Optional Boolean parameter, sets the analyzer to internal (OFF) or external (ON) reference frequency. |
| *RST values | *RST has no effect on the configured generators. |
| **SCPI, Command Types** | Device-specific, command or query |
| **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. |

## SYSTem:COMMunicate:RDEVice:GENerator<gen_no>:DELete

Clears the configuration table for external generators.

| | |
|---|---|
| **<gen_no>** | Number of the configured generator. This number is ignored, the command clears all entries in the configuration table. |
| **SCPI, Command Types** | Device-specific, no query |
| **Example:** | `See SYSTem:COMMunicate:RDEVice:GENerator<gen_no>:DEFine` |

## SYSTem:COMMunicate:RDEVice:PMETer<pmeter_no>:DEFine '<pmeter_name>', '<driver>', '<interface>', '<address>'

Configures an external power meter and adds it to the list of available power meters.

| | |
|---|---|
| **<pmeter_no>** | 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.<br>Generators can be assigned several times so that the number of configured power meters is practically unlimited. |
| **'<pmeter_name>'** | 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.<br>**Alternative:** The '<pmter_name>' string may contain the power meter driver file name with its complete path, e.g. 'C:\Program Files\Rohde&Schwarz\Network Analyzer\resources\extdev\nrvd.pwm'. |
| **'<interface>'** | Interface type (string parameter): 'USB' (for the supported USB devices) |
| **'<address>'** | Interface address (string parameter), depending on the interface type. |
| *RST values | *RST has no effect on the configured power meters. |
| **SCPI, Command Types** | Device-specific, command or query |
| **Example:** | `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. |
| | `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. |

## SYSTem:COMMunicate:RDEVice:PMETer<pmeter_no>:DELete

Clears the configuration table for external power meters.

| | |
|---|---|
| **<pmeter_no>** | Number of the configured power meter. This number is ignored, the command clears all entries in the configuration table. |
| **SCPI, Command Types** | Device-specific, no query |
| **Example:** | `See SYSTem:COMMunicate:RDEVice:PMETer<pmeter_no>:DEFine` |

## SYSTem:DATA:SIZE ALL | AUTO

Defines the block size for the data transfer between the instrument hardware and the software. The block size has an impact on the display of traces and on the readout of trace data using the commands of the `TRACe...` or `CALCulate<Chn>:DATA...` subsystems.

| | |
|---|---|
| **ALL** | Data transfer with full buffer size. The data for each sweep are transferred together so that each displayed trace contains the data of exactly one sweep. The same is true for queried trace data. The trace is displayed or updated at once, the sweep progress is not visible on the screen.<br> To make sure that all queried trace points stem from the same sweep you can also use single sweep mode (`INITiate<Ch>:CONTinuous OFF`) and the common command *OPC? |
| **AUTO** | Data transfer with an automatically determined block size. In general the data blocks comprise only part of a trace, therefore the sweep progress is visible on the screen. On the other hand the displayed trace can show the results of two consecutive sweeps. |
| *RST value | AUTO. |
| **SCPI, Command Types** | Device-specific, command or query |
| **Example:** | `SYST:DATA:SIZE ALL`<br>Select data transfer with full buffer size in order to view the trace data from a single sweep.. |

## SYSTem:DISPlay:COLor DBACkground | LBACkground | BWLStyles | BWSolid

Selects the color scheme for all diagram areas in the active setup.

| | |
|---|---|
| **DBACkground** | Dark background |
| **LBACkground** | Light background |
| **BWLStyles** | Black and white line styles |
| **BWSolid** | Black and white solid |
| *RST value | –<br>*RST does not affect the color settings; see also description of the Preset command. |
| **SCPI, Command Types** | Device-specific, command or query |
| **Example:** | `SYST:DISP:COL LBAC`<br>Select a light background, e.g. to generate color hardcopies. |

## SYSTem:DISPlay:UPDate <Boolean> | ONCE

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.

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`.

| | |
|---|---|
| **<Boolean>** | 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. |
| *RST value | OFF |
| **SCPI, Command Types** | Device-specific, command or query |
| **Example:** | `SYST:DISP:UPD ON` |
| | Switch the display on to view the traces and diagrams. |

## 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.

| | |
|---|---|
| ***RST value** | – |
| **SCPI, Command Types** | Confirmed, query only |
| **Example:** | `SYST:ERR?` |
| | Query the oldest entry in the error queue. 0,"No error" is returned if the error queue is empty. |

## 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.

| | |
|---|---|
| ***RST value** | – |
| **SCPI, Command Types** | Confirmed, command or query |
| **Example:** | `SYST:ERR:ALL?` |
| | Query all entries in the error queue. 0,"No error" is returned if the error queue is empty. |

## SYSTem:ERRor:DISPlay <Boolean>

Switches the display of a tooltip for remote command errors on or off. The tooltip appears at the bottom of the remote screen and of the manual screen; it is **not** displayed for SCPI errors no. –113, *Undefined header.*

| | |
|---|---|
| **<Boolean>** | ON \| OFF – Switch the tooltip on or off. |
| *RST value | OFF |
| **SCPI, Command Types** | Device-specific, command or query |
| **Example:** | `SYST:ERR:DISP ON` |
| | Switch the display of the tooltip for remote command errors on. |
| | `FREQ:STAR 1` |
| | Remote Error : -222,"Data out of range;FREQ:STAR 1" |

## SYSTem:FIRMware:UPDate '<file_name>'

Installs a firmware version stored in a NWA setup file (*.msi) on the analyzer. The installation is automatic and does not require any further action to be taken.

| | |
|---|---|
| **'<file_name>'** | String variable for the name and directory of a NWA setup file (*.msi). |
| *RST value | – |
| **SCPI, Command Types** | Device-specific, no query |
| **Example:** | `SYST:FIRM:UPD 'C:\Setup\ZVAB_01.10.msi'` |
| | Install the firmware version 01.10 from the setup file stored in the *Setup* directory of the analyzer's internal hard disk. |

## SYSTem:FPReset

Performs a factory preset of all instrument settings (i.e. all open setups) or of the active setup, depending on the `SYSTem:PRESet:SCOPe` settings, and deletes channel 1 in the active setup. As a result, the active setup contains no channels traces, and diagram areas.

| | |
|---|---|
| *RST value | – |
| **SCPI, Command Types** | Device-specific, no query |
| **Example:** | `SYST:PRES:SCOP SING` |
| | Define the scope of a preset: the active setup is reset only. |
| | `SYST:FPR` |
| | Reset the parameters of the current setup and delete channel 1. |

## SYSTem:KLOCk <Boolean>

Locks or unlocks the local controls of the analyzer. This includes the front panel keys, the keyboard, or other local interfaces.

| | |
|---|---|
| **<Boolean>** | ON \| OFF – Lock or unlock the local keys. |
| *RST value | OFF |
| **SCPI, Command Types** | Confirmed, command or query |
| **Example:** | `SYST:KLOCk ON` |
| | Lock the local keys. |

## 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. Refer to the service manual for more information.

| | |
|---|---|
| **'<password>'** | Case-sensitive string variable. Sending an invalid password generates error -221, (settings conflict). |
| *RST value | – (neither the password nor the protection of service functions is affected by *RST) |
| **SCPI, Command Types** | Confirmed, no query |
| **Example:** | `SYST:PASS "XXXX"` |
| | Enter password. |

## SYSTem:PRESet

Performs a factory preset of all instrument settings (i.e. all open setups) or of the active setup, depending on the `SYSTem:PRESet:SCOPe` settings. The command is equivalent to *RST and to the action of the *PRESET* key on the front panel.

*If a user-defined preset is active (`SYSTem:PRESet:USER[:STATe] ON`), the PRESET key initiates a user-defined preset, while `SYSTem:PRESet` and `*RST` still activate the factory preset.*

| | |
|---|---|
| *RST value | – |
| **SCPI, Command Types** | Confirmed, no query |
| **Example:** | `SYST:PRES:SCOP SING` |
| | Define the scope of a preset: the active setup is reset only. |
| | `SYST:PRES` |
| | Reset the parameters of the current setup. |

## SYSTem:PRESet:SCOPe ALL | SINGle

Specifies whether a preset (SYSTem:PRESet; `*RST`) affects the active setup only or all open setups.

| | |
|---|---|
| **ALL** | All open setups are deleted and the setup *Set1* is created with default trace and channel settings. |
| **SINGle** | The settings of the active setup are reset; the name of the active setup and the parameters of all other setups remain unchanged. |
| *RST value | – (*RST does not affect the preset scope setting) |
| **SCPI, Command Types** | Device-specific, command or query |
| **Example:** | `See SYSTem:PRESet` |

## SYSTem:PRESet:USER[:STATe] <Boolean>

Selects a factory preset or a user-defined preset.

*The user-defined preset can be initiated using System – Preset (manual control) or `MMEMory:LOAD:STATe. *RST` and `SYSTem:PRESet` always initiate a factory preset.*

| | |
|---|---|
| **<Boolean>** | OFF - User-defined preset switched off (factory preset is used)<br>ON - User-defined preset switched on |
| *RST value | – (*RST does not affect the preset state setting) |
| **SCPI, Command Types** | Device-specific, command or query |
| **Example:** | `SYST:PRES:USER ON`<br>Enable a user-defined preset.<br>`SYST:PRES:USER:NAME 'C:\Rohde&Schwarz\Nwa\RecallSets\Setup_2.zvx'`<br>Select a setup file for the user-defined preset.<br>Press *System – Preset* (manual control) **or** use<br>`MMEM:LOAD:STAT 1, 'C:\Rohde&Schwarz\Nwa\RecallSets\Setup_2.zvx'`<br>Carry out the use-defined preset. |

## SYSTem:PRESet:USER:NAME '<Setup_file>'

Specifies the name of a setup file (.zvx) to be used for a user-defined preset.

| | |
|---|---|
| **'<Setup_file>'** | String parameter to specify the name and directory of the setup file to be loaded. The default extension (manual control) for setup files is *.zvx, although other extensions are allowed. If no path is specified the analyzer searches the default directory (subdirectory `...RecallSets`) |
| *RST value | – (*RST does not affect the preset settings) |
| **SCPI, Command Types** | Device-specific, command or query |
| **Example:** | `See SYSTem:PRESet:USER[:STATe]` |

## SYSTem:SETTings:UPDate ONCE

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`).

| | |
|---|---|
| **ONCE** | Causes an immediate update of the settings. |
| *RST value | – |
| **SCPI, Command Types** | Confirmed, no query |
| **Example:** | `INIT:CONT OFF`<br>`Activate single sweep mode.`<br>`SYST:SETT:UPD ONCE` |
| | Update the settings made during the current single sweep period. |

## SYSTem:SOUNd:ALARm[:STATe] <Boolean>

Switches alarm sounds on or off.

| | |
|---|---|
| **<Boolean>** | OFF - Alarm sounds switched off<br>ON - Alarm sounds switched on |
| *RST value | ON |
| **SCPI, Command Types** | Device-specific, command or query. |
| **Example:** | `SYST:SOUN:ALAR OFF; STAT OFF` |
| | Switch alarm and status sounds off. |

## SYSTem:SOUNd:STATus[:STATe] <Boolean>

Switches status sounds on or off.

| | |
|---|---|
| **<Boolean>** | OFF - Status sounds switched off<br>ON - Status sounds switched on |
| *RST value | ON |

**SCPI, Command Types** Device-specific, command or query.

**Example:**
```
See SYSTem:SOUNd:ALARm[:STATe]
```

## SYSTem:USER:DISPlay:TITLe '<title>'

Defines a title for the remote display.

**'<title>'** Title (string variable)
*RST value – (*RST does not affect the title)

**SCPI, Command Types** Device-specific, command or query.

**Example:**
```
SYST:USER:DISP:TITL 'Remote test running'
```
Define a title for the remote display.

## SYSTem:USER:KEY <ukey_no>[, '<ukey_name>']

Labels a user-defined key in the remote display. In the query form the command returns whether or not a user-defined key was used.

**<ukey_no>** Number of the user key
Range [def. unit] 0 – Delete all user keys and restore the default keys *(Go to Local, Display Off)*
1 to 8 – user key numbers
*RST value 0

**'<ukey_name>'** Label for user key no. 1 to 8 (string variable)
*RST value " (empty string)

**SCPI, Command Types** Device-specific, command or query.

**Example:**
```
SYST:USER:KEY 1, 'User S11'
```
Define a user key no. 1 labeled S11. The user key is only labeled, no functionality has been assigned yet.

```
SYST:USER:KEY:FUNC 1, 'S11'
```
Assign the functionality 'S11' (select S11 as a measured quantity for the active trace) to the created user key.

```
SYST:USER:KEY?
```
The query returns *0,",* indicating that no user key has been pressed. If you press the user softkey no. 1, the response is *1, 'User S11'.*

```
CALC:PAR:MEAS? 'Trc1'
```
The query returns *'S11',",* indicating that the measured quantity for trace 'Trc1' has been changed.

```
SYST:USER:KEY 0
```
Delete the user key and restore the default keys..

### SYSTem:USER:KEY:FUNCtion <ukey_no>[, '<ukey_name>']

Labels the functionality of a function key to a user-defined key in the remote display.

You can use this command to execute part of your measurement task manually; see *Combining Manual and Remote Control*.

| | |
|---|---|
| **<ukey_no>** | Number of the user key |
| Range [def. unit] | 1 to 8 – user key numbers |
| *RST value | – (the parameter is also needed in the query form) |
| **'<ukey_name>'** | Name of a function key, e.g. 'S11', 'Start' etc. (string variable) |
| *RST value | '' (empty string) |
| **SCPI, Command Types** | Device-specific, command or query. |
| **Example:** | `See SYST:USER:KEY` |

### SYSTem:VERSion?

Returns the SCPI version number to which the analyzer complies. The analyzer complies to the final SCPI version 1999.0

| | |
|---|---|
| ***RST value** | – |
| **SCPI, Command Types** | Confirmed, query only |
| **Example:** | `SYST:VERS?` |
| | Query the SCPI version. The response is *1999.0*. |

## TRACe...

This subsystem handles active trace data and trace data stored in the analyzer's internal memory.

*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`).*

### Reserved Trace Names

The commands in the `TRACe...` menu use the following ZVR-compatible parameters to specify traces:

| Parameter | Meaning | Used in |
|---|---|---|
| **CH1DATA, CH2DATA, CH3DATA, CH4DATA** | Active data trace of channels 1 to 4 | `TRACe:COPY`<br>`TRACe[:DATA][:RESPonse][:ALL]?`<br>`TRACe[:DATA][:STIMulus][:ALL]?`<br>`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][:RESPonse][:ALL]?`<br>`TRACe[:DATA][:STIMulus][:ALL]?` |
| **IMPLied** | Active data trace, addressed with | `CALCulate<Chn>:MATH[:EXPRession][:DEFine]` |

| | <Chn> | |
|---|---|---|
| **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`<br>`TRACe:COPY`<br>`TRACe[:DATA][:RESPonse][:ALL]?`<br>`TRACe[:DATA][:STIMulus][:ALL]?`<br>`CALCulate<Chn>:MATH[:EXPRession][:DEFine]` |

## TRACe:CLEar MDATA1 | MDATA2 | MDATA3 | MDATA4 | MDATA5 | MDATA6 | MDATA7 | MDATA8

Deletes one of the memory traces *Mem<n>[Trc<m>],* where n = 1, ... 8.

**Parameters** Identifier for the memory trace; see list of trace names. .
Range [def. unit] MDATA<n> where <n> = 1 to 8. [–]
*RST value –

**SCPI, Command Types** Device-specific, no query

**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).

## TRACe:COPY <memory_trc>,<data_trc>

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.

*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.*

**<memory_trc>** Name of the memory trace.

Range [def. unit] <memory_trace> is either a string variable (enclosed in single or double quotes) or one of the following reserved names (no string variables):
ZVR MDATA1 | MDATA2 | MDATA3 | MDATA4 | MDATA5 | MDATA6 | MDATA7 | MDATA8 (only for memory traces *Mem<n>[Trc<m>],* where n = 1, ... 8; see list of trace names). [–]

*RST value –

**<data_trc>** Name of the data trace

Range [def. unit] <data_trace> is either a string variable (enclosed in single or double quotes) or one of the following reserved names (no string variables):
ZVR CH1DATA | CH2DATA | CH3DATA | CH4DATA (only for the active data trace in channels Ch1, Ch2, Ch3, Ch4; see list of trace names). [–]

*RST value –

**SCPI, Command Types** Confirmed, no query

**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).

## TRACe:COPY:MATH <memory_trc>,<data_trc>

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.*

**<memory_trc>** Name of the memory trace.

Range [def. unit] <memory_trace> is either a string variable (enclosed in single or double quotes) or one of the following reserved names (no string variables):

ZVR ☑MDATA1 | MDATA2 | MDATA3 | MDATA4 | MDATA5 | MDATA6 | MDATA7 | MDATA8 (only for memory traces *Mem<n>[Trc<m>],* where n = 1, ... 8; see list of trace names). [–]

*RST value –

**<data_trc>**
Range [def. unit]

Name of the data trace
<data_trace> is either a string variable (enclosed in single or double quotes) or one of the following reserved names (no string variables):
ZVR ☑CH1DATA | CH2DATA | CH3DATA | CH4DATA (only for the active data trace in channels Ch1, Ch2, Ch3, Ch4; see list of trace names). [–]

*RST value –

**SCPI, Command Types** Confirmed, no query

**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.

## TRACe[:DATA][:RESPonse][:ALL]? CH1DATA | CH2DATA | CH3DATA | CH4DATA | CH1MEM | CH2MEM | CH3MEM | CH4MEM | MDATA1 | MDATA2 | MDATA3 | MDATA4 | MDATA5 | MDATA6 | MDATA7 | MDATA8

Returns the response values of the active data trace or memory trace (see trace names).

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?*

**<Response>**
Range [def. unit]

Response data of the selected trace, see list of trace names.
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.

*RST value –

**SCPI, Command Types** Device-specific, query only

**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.

### TRACe[:DATA]:STIMulus[:ALL]? CH1DATA | CH2DATA | CH3DATA | CH4DATA | CH1MEM | CH2MEM | CH3MEM | CH4MEM | MDATA1 | MDATA2 | MDATA3 | MDATA4 | MDATA5 | MDATA6 | MDATA7 | MDATA8

Returns the stimulus values of the active data trace or memory trace (see trace names).

*To read the stimulus values of an arbitrary data or memory trace, use CALCulate<Chn>:DATA:STIMulus?*

| | |
|---|---|
| **<Response>** | Response data of the selected trace, see list of trace names. |
| Range [def. unit] | The data is transferred in the data format defined via FORMat[:DATA]. The numeric values are expressed in the default unit of the stimulus variable. [Hz, dBm or s, depending on the sweep type] |
| *RST value | – |
| **SCPI, Command Types** | Device-specific, query only |
| **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.

## TRIGger<Ch>...

This subsystem is used to synchronize the analyzer's actions (in particular, the start of a measurement sequence) with events.

### TRIGger<Ch>[:SEQuence]:HOLDoff <numeric_value>

Sets a delay time between the trigger event and the start of the measurement *(Trigger Delay)*.

| | |
|---|---|
| **<Ch>** | Channel number. If unspecified the numeric suffix is set to 1. |
| **<numeric_value>** | Delay time. |
| Range [def. unit] | 0 s to 13680 s [s]. UP and DOWN increment/decrement the trigger delay in 10 ms steps. |

| | |
|---|---|
| *RST value | 0 s |
| **SCPI, Command Types** | Confirmed, command or query |
| **Example:** | `TRIG:SOUR TIM` |
| | Activate internal timer as trigger source. |
| | `TRIG:HOLD UP` |
| | Set a delay time of 10 ms. |

## TRIGger<Ch>[:SEQuence]:LINK 'POINt' | 'SWEep' | 'PPOint' | 'SEGMent'

Selects the *Triggered Meas. Sequence*. The identifier for the sequence is a string variable.

| | |
|---|---|
| **<Ch>** | Channel number. |
| **'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 value | 'SWEep' |
| **SCPI, Command Types** | Confirmed, command or query. The query returns the complete string; see example below. |
| **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'. |

## TRIGger<Ch>[:SEQuence]:SLOPe POSitive | NEGative

Qualifies whether the trigger event occurs on the rising or on the falling edge of the external TTL trigger signal.

| | |
|---|---|
| **<Ch>** | Channel number. |
| **POSitive \|** | Rising edge. |
| **NEGative** | Falling edge. |
| *RST value | POSitive |
| **SCPI, Command Types** | Confirmed, command or query |
| **Example:** | `TRIG:SOUR EXT` |

Activate external signal as trigger source.

```
TRIG:SLOP NEG
```

Trigger on the negative edge of the (external TTL) trigger signal.

## TRIGger<Ch>[:SEQuence]:SOURce IMMediate | EXTernal | TIMer | MANual

Selects the source for the events that the analyzer uses to start a sweep.

| | |
|---|---|
| **<Ch>** | Channel number. |
| **IMMediate** | Free run measurement without waiting for trigger events |
| **EXTernal** | Trigger by external signal applied to pin 2 of the *USER CONTROL* connector on the rear panel. |
| **TIMer** | Periodic internal trigger signal. The period of the timer is specified by the `TRIGger[:SEQuence]:TIMer` command. |
| **MANual** | Trigger signal generated by pressing the *Manual Trigger* softkey. |
| *RST value | IMMediate |
| **SCPI, Command Types** | Confirmed, command or query. |
| **Example:** | `TRIG:SOUR MAN` |
| | Activate manual trigger mode. The analyzer starts the next sweep when the *Manual Trigger* softkey is pressed. |

## TRIGger<Ch>[:SEQuence]:TIMer <numeric_value>

Sets the period of the internal periodic signal that can be used as a trigger source (`TRIGger<Ch>[:SEQuence]:SOURce TIMer`).

| | |
|---|---|
| **<Ch>** | Channel number. If unspecified the numeric suffix is set to 1. |
| **<numeric_value>** | Timer period |
| Range [def. unit] | 1E-005 s to 13680 s [s]. UP and DOWN increment/decrement the timer period in 10 ms steps. |
| *RST value | 1 s |
| **SCPI, Command Types** | Confirmed, command or query |
| **Example:** | `TRIG:SOUR TIM` |
| | Activate internal timer as trigger source. |
| | `TRIG:TIM 0.1` |
| | Set a timer period of 100 ms. |

# Contents of Chapter 7

7 Program Examples ..... 1

**Typical Stages of a Remote Control Program ..... 1**

Basic Instrument Settings ..... 1

Adjusting the Test Setup ..... 2

Start of the Measurement and Command Synchronization ..... 2

Retrieving Measurement Results ..... 3

**Handling of Channels, Traces and Diagram Areas ..... 4**

Several Traces with Equal Channel Settings ..... 4

Several Traces with Different Channel Settings... ..... 9

**Markers and Limit Lines... ..... 12**

**Condensed Programming Examples ..... 17**

**Calibration ..... 17**

**Modeling a Max Hold Function ..... 18**

**Retrieving the Results of Previous Sweeps ..... 19**

# 7 Program Examples

This chapter contains detailed program examples on the following subjects:

- Traces, Channels and Diagram Areas
- Calibration

The syntax and use of all SCPI commands is described in the *SCPI Reference* chapter. For a general introduction to remote control of the analyzer refer to chapter *Remote Control*.

# Typical Stages of a Remote Control Program

A typical remote control program comprises the following stages:

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.*

## 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 discussed in section *Optimizing the Measurement Speed.* 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...

## 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. in order to replace the DUT, change the connected ports, connect external devices etc.

## 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]` is used to start a single sweep or a group of single sweeps. This command has been implemented for overlapped execution. The advantage of an overlapped command 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 command),

// 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

// on the GPIB bus

INITiate1:SCOPe SINGle // Single sweep will be started in the referenced channel only

INITiate1:IMMediate; *WAI // Start single sweep in channel no. 1, wait until the end of the sweep

<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.

INITiate1:IMMediate; *OPC? // Start single sweep in channel no. 1, indicate the end of the sweep by a 1 in the output queue.

// So far the controller may still send messages to other devices on the GPIB bus.

<Condition OPC=1> // Stop the controller until *OPC? returns one (program syntax depends on your programming environment).

<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.

*SRE 32 / Enable a service request for the ESR

*ESE 1 / Set event enable bit for operation complete bit

INITiate1::IMMediate; *OPC // 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.

<Wait for service request> // Controller waits for service request from the analyzer (program syntax depends on your programming environment).

<Continue program sequence>

## 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` to retrieve a particular trace within a group of sweeps.

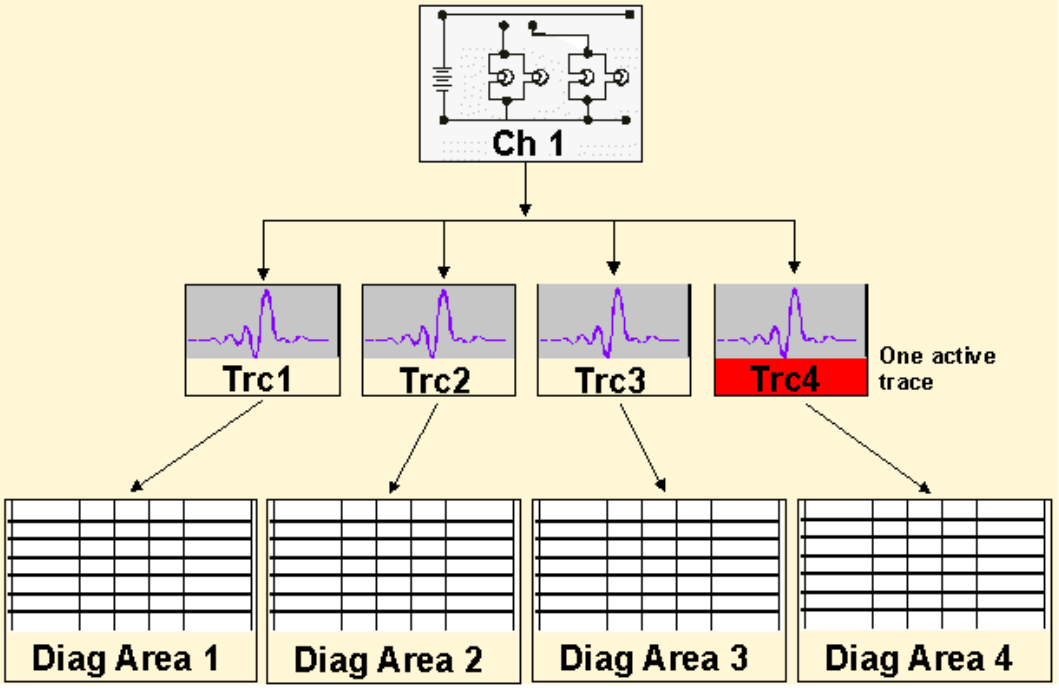
# Handling of Channels, Traces and Diagram Areas

The following examples show you how to perform basic tasks related to channel and trace definition and to the display of traces in diagram areas.

*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.*

## 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 diagram areas.




**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.
- Diagram areas 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 area.

- In remote control, it is possible to display the same trace in several diagram areas.
- The analyzer provides several commands allowing a smooth transition between remote and manual control.

//

**// 1. One channel, two traces, one diagram area**

// 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 area.

CALCulate1:PARameter:SDEFine 'Trc2', 'S21' // the trace becomes the active trace but is not displayed

CALCulate1:FORMat PHASe // the trace is referenced by the channel suffix 1

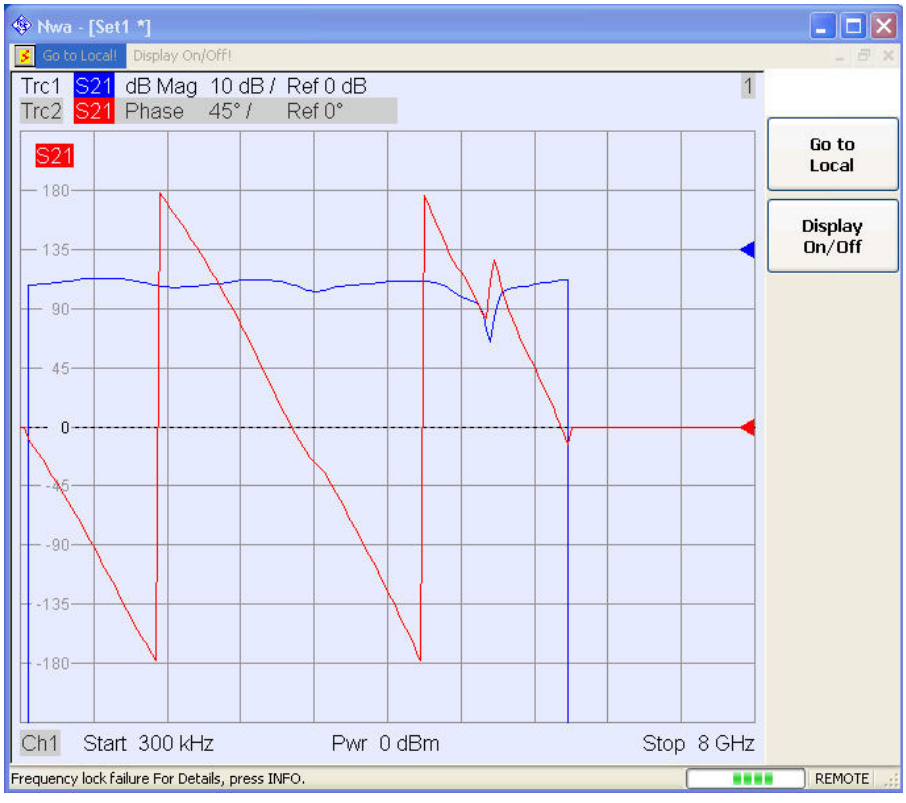
DISPlay:WINDow1:TRACe2:FEED 'Trc2' // display the second trace, numbering it the second trace in diagram area no. 1

//

// **Check the result on the local screen**

// Go to local

SYSTem:DISPlay:UPDate ONCE




//

//

**// 2. One channel, two traces, two diagram areas**

// Create a second diagram area, 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 diagram areas

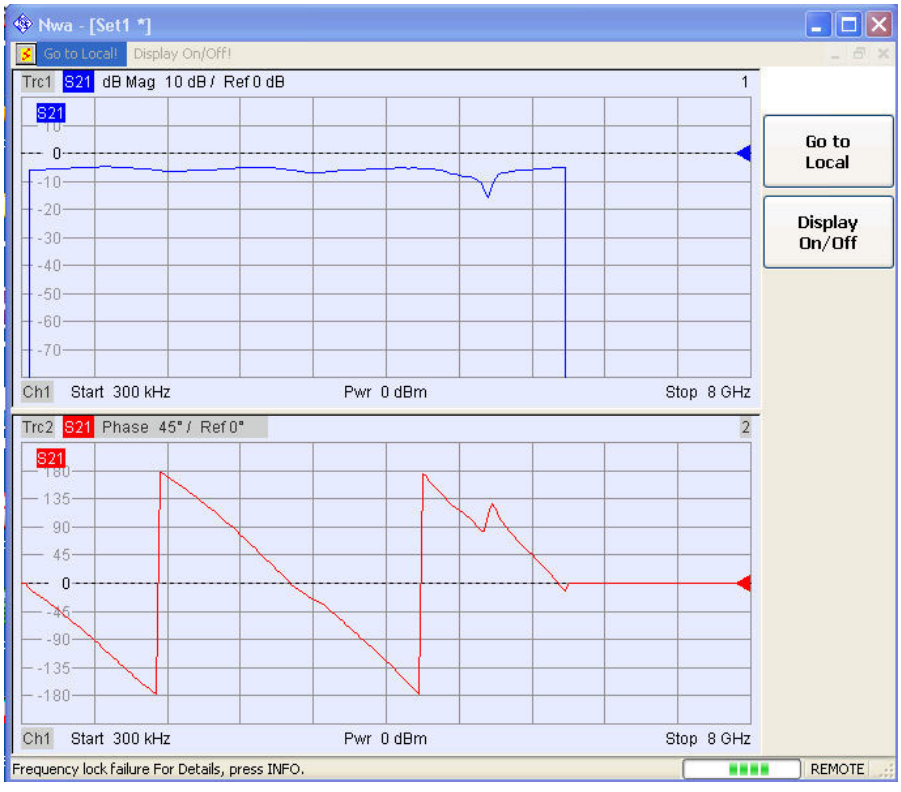
DISPlay:WINDow1:TRACe2:DELete

//

// **Check the result on the local screen**

// Go to local

SYSTem:DISPlay:UPDate ONCE




//

**// 3. One channel, four traces, four diagram areas**

// Reset the instrument, add diagram areas 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'

CALCulate1:FORMat SMITh // Smith chart for the active trace Trc4, referenced by the channel number

//

// Display the new traces in diagram areas no. 2 to 4.

DISPlay:WINDow2:TRACe2:FEED 'Trc2'

DISPlay:WINDow3:TRACe3:FEED 'Trc3'

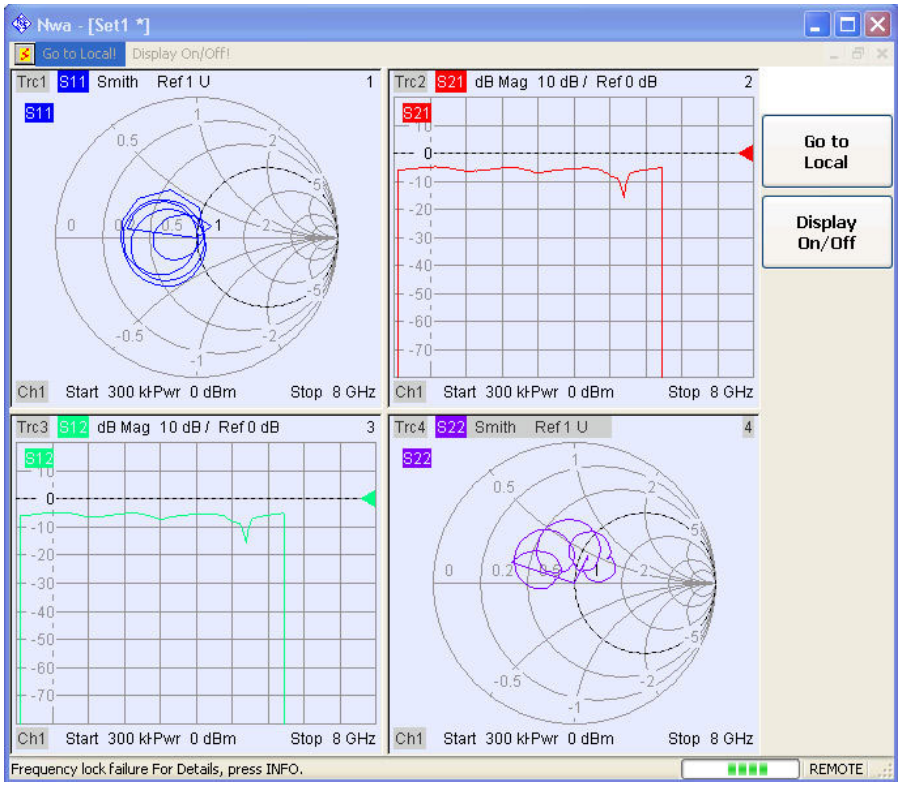
DISPlay:WINDow4:TRACe4:FEED 'Trc4'

//

// **Check the result on the local screen**

// Go to local

SYSTem:DISPlay:UPDate ONCE

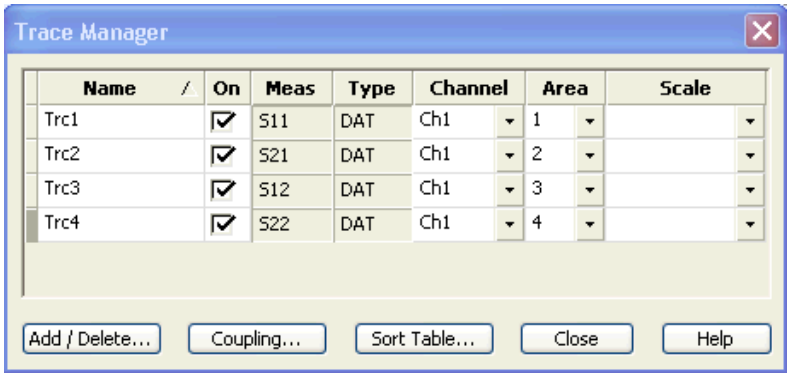



//

// **Check the result in the trace manager**

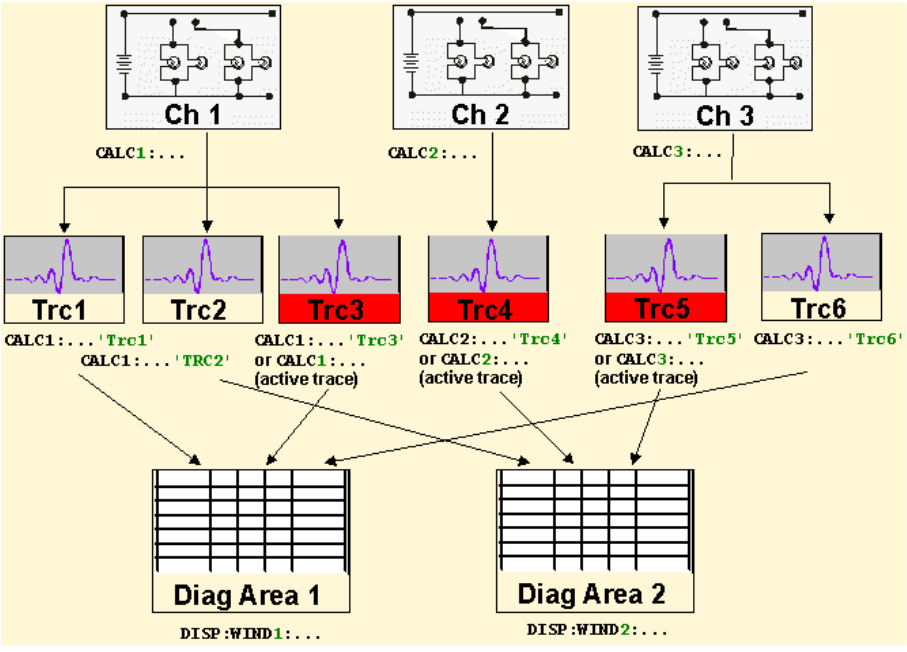
// The trace manager gives an overview of the current channel/trace configuration

DISPlay:MENU:KEY:EXECute 'Trace Manager'



# Several Traces with Different Channel Settings...

**Programming task:** Create three channels with 3, 1 and 2 traces, respectively, and display the traces in two diagram areas.




**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:

- Channels are always referenced by a channel suffix.
- Traces are referenced by trace names. The active trace of a channel is often referenced by the channel suffix.
- Diagram areas 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 area.
- 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

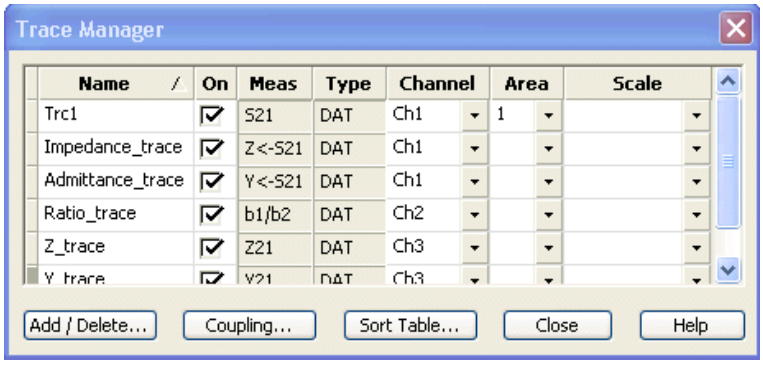
// So far, only the default trace is displayed.

//

// **Check the result in the trace manager**

// The trace manager gives an overview of the current channel/trace configuration

DISPlay:MENU:KEY:EXECute 'Trace Manager'

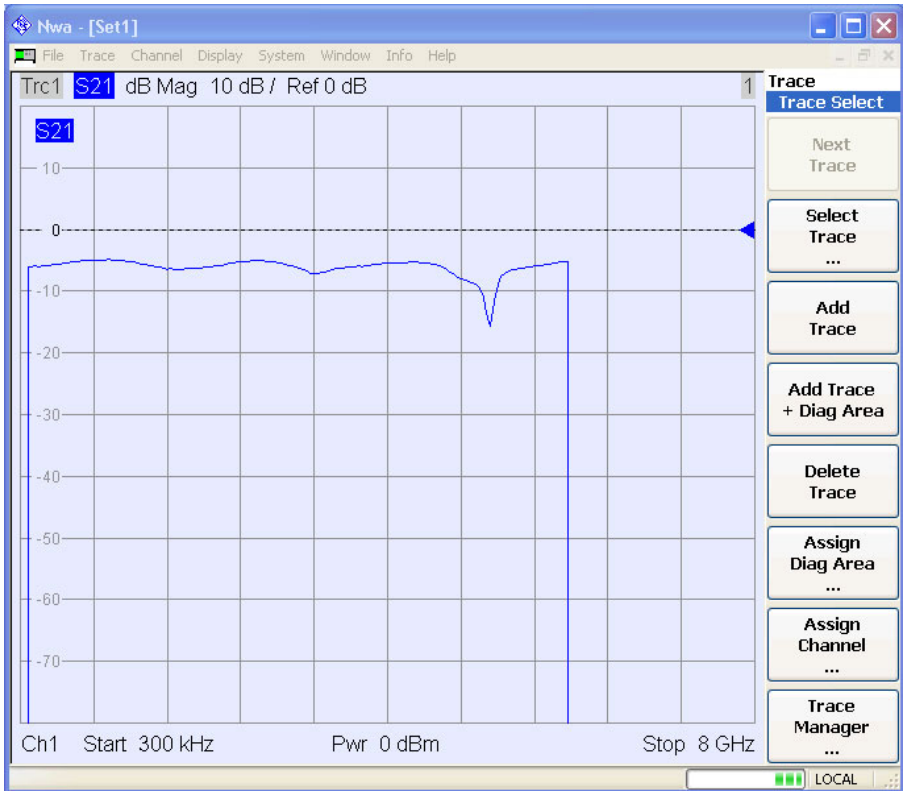



//

// **Check the result on the local screen**

// Go to local

SYSTem:DISPlay:UPDate ONCE




//

**// 2. Create second diagram area 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'

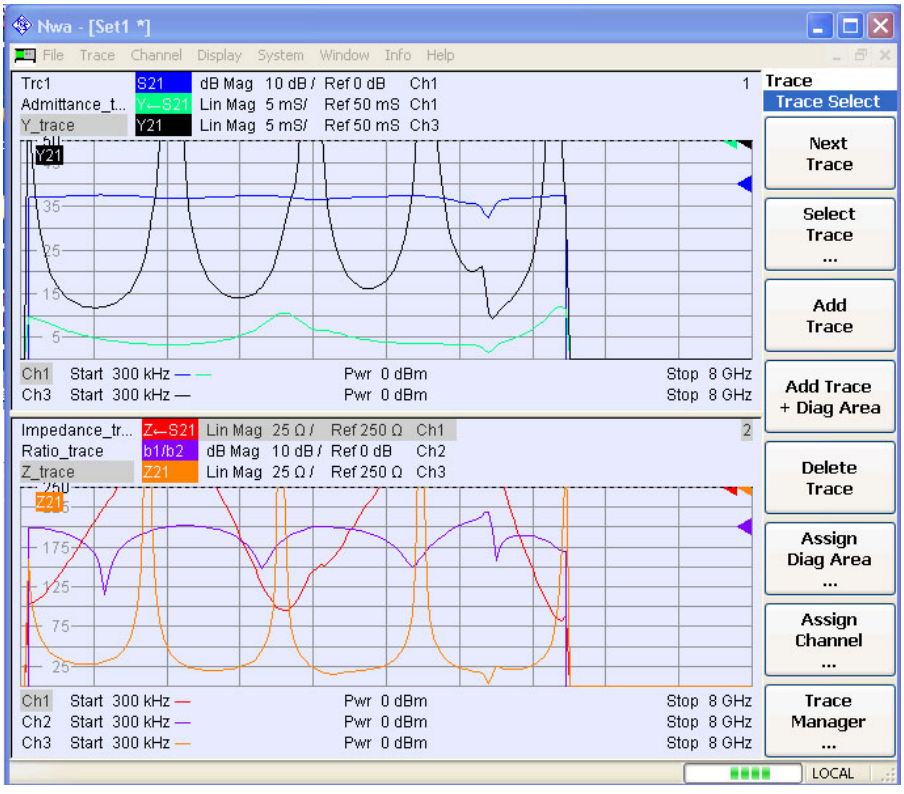
DISPlay:WINDow2:TRACe3:FEED 'Z_trace'

//

// **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 area no. 2.

DISPlay:WINDow2:TRACe3:Y:RLEVel?

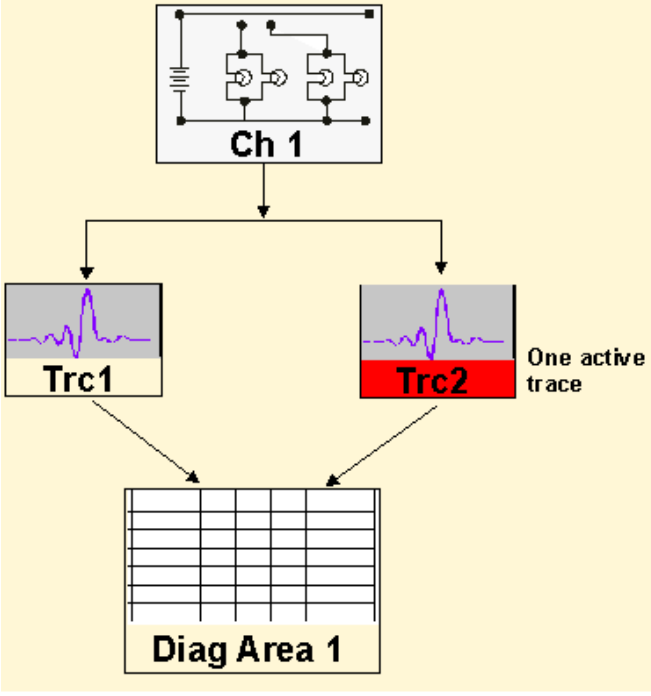
//

// 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.

CALCulate3:FORMat PHASe

# Markers and Limit Lines...

**Programming task:** Display two traces in a single diagram ares, 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:

- 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.
- Diagram areas 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 area.
- The analyzer provides several commands allowing a smooth transition between remote and manual control.

//

**// 1. Create one channel, two traces, one diagram area**

// 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 area.

CALCulate1:PARameter:SDEFine 'Trc2', 'S21' // the trace becomes the active trace but is not displayed

CALCulate1:FORMat PHASe // the trace is referenced by the channel suffix 1

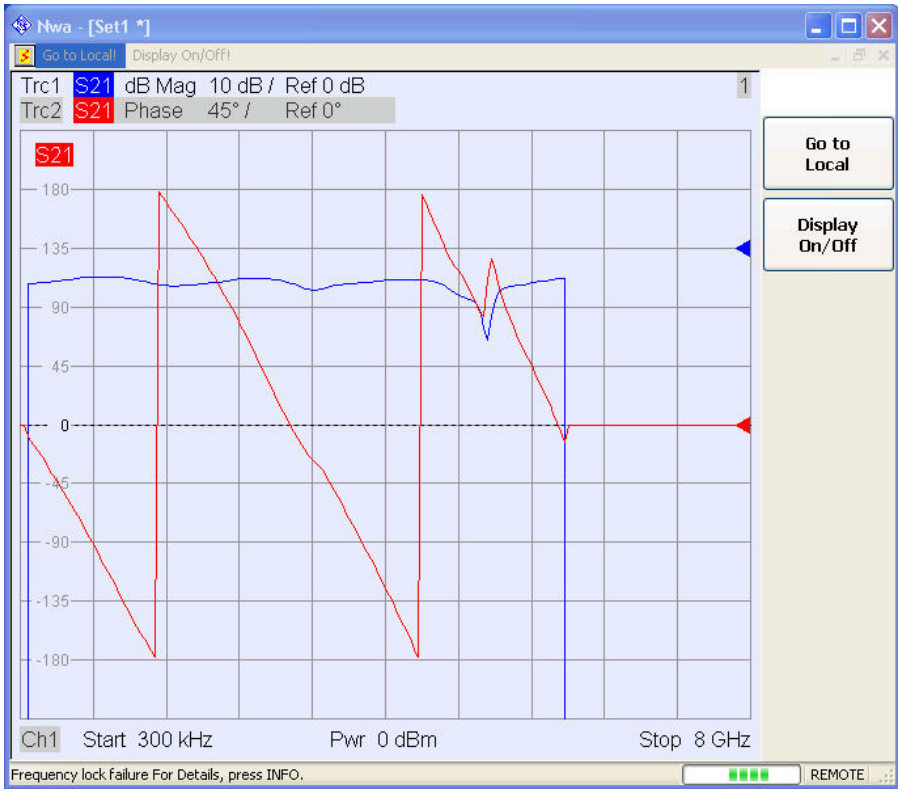
DISPlay:WINDow1:TRACe2:FEED 'Trc2' // display the second trace, numbering it the second trace in diagram area no. 1

//

// **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

DISPlay:WINDow1:TRACe1:Y:SCALe:AUTO ONCE / in the autoscale command the trace is referenced by its number in the diagram

//

// 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'

CALCulate1:MARKer1:STATe ON // the marker is set to the center of the sweep range

CALCulate1:MARKer1:DELTa:STATe ON // this command also creates the reference marker

CALCulate1:MARKer1:REFerence:X 4.5 GHz // set the reference marker to the beginning of the sweep range

//

// Use the delta marker to search for the minimum of the trace and query the result.

CALCulate1:MARKer1:FUNCtion:EXECute MIN; RES? // the query returns the stimulus and the

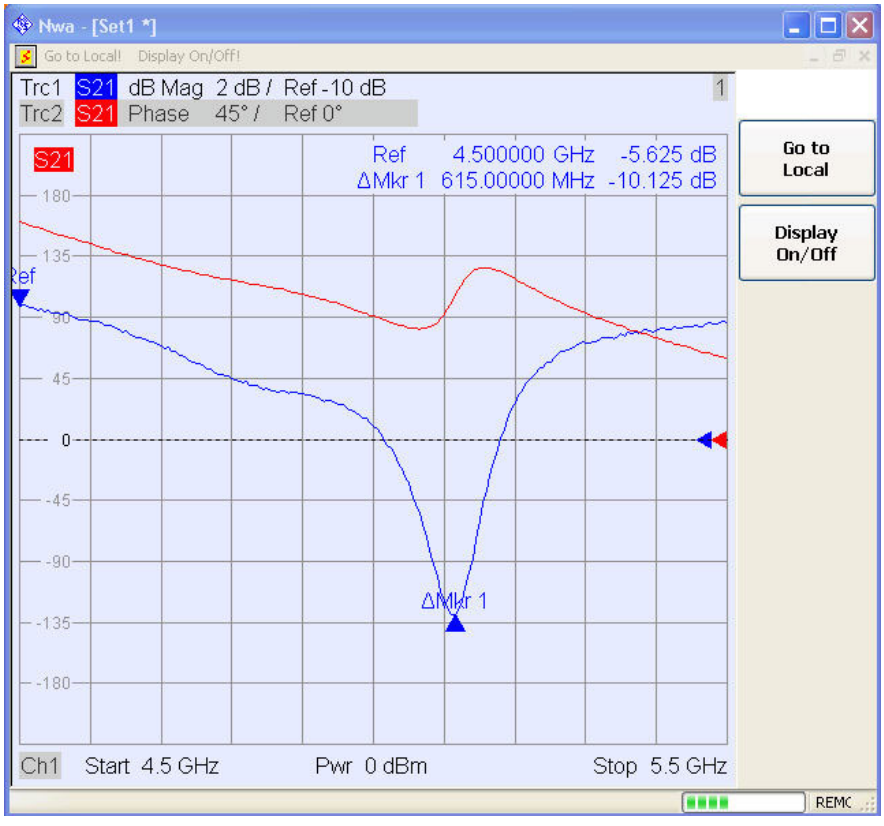
response value at the marker position

//

// **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.*

//

//

**// 2. Limit lines and limit check**

// Remove all markers and define a limit line for the active trace.

CALCulate1:MARKer1:AOFF

CALCulate1:LIMit:DATA 1, 4500000000, 5500000000, -5, -5 // define an upper limit line across the entire sweep range

CALCulate1:LIMit:DATA 2, 4500000000, 5000000000, -10, -15

CALCulate1:LIMit:DATA 2, 5000000000, 5500000000, -15, -10 // define two segments for the lower limit line

//

// Display the limit line and perform the limit check.

CALCulate1:LIMit:DISPlay:STATe ON

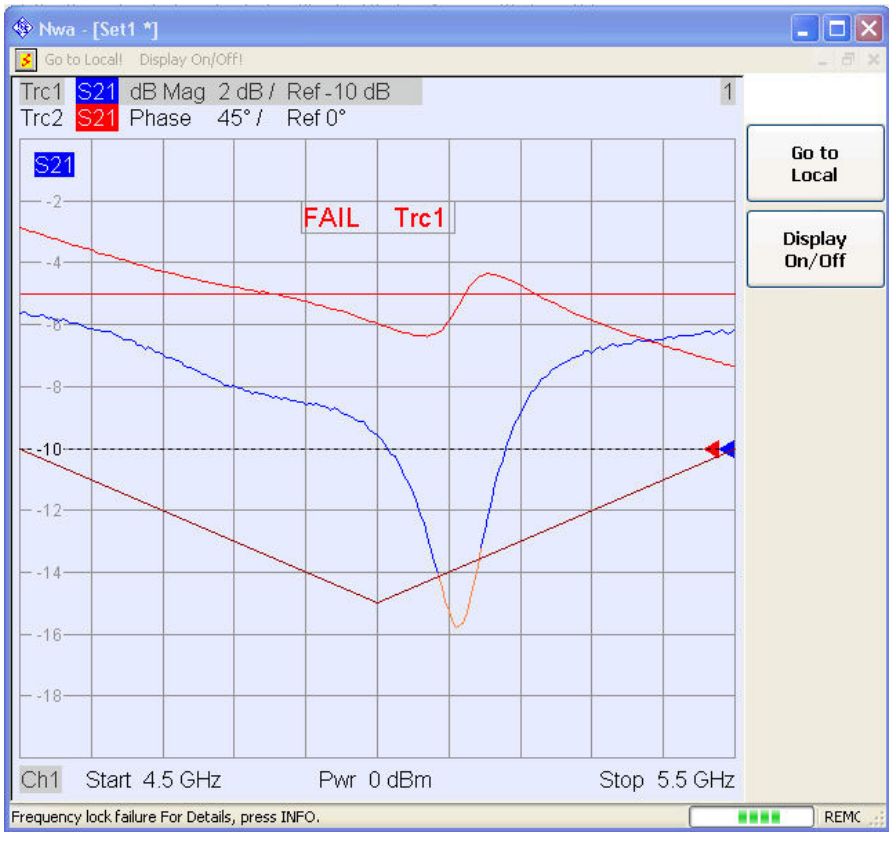
CALCulate1:LIMit:STATe ON; FAIL? / if the trace is failed; the response is 1.

//

// **Check the result on the local screen**

// Go to local

SYSTem:DISPlay:UPDate ONCE

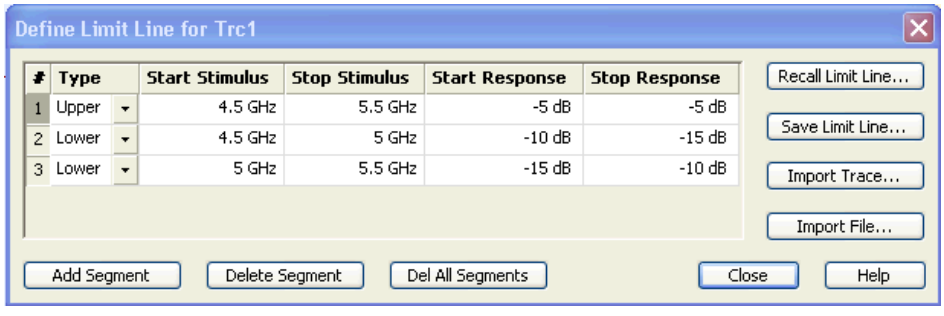



//

// **Check the result in the *Define Limit Line* dialog**

// The *Define Limit Line* dialog gives an overview of the limit line segments of the active trace.

DISPlay:MENU:KEY:EXECute 'Define Limit Line'



# Condensed Programming Examples

# Calibration

The following example shows you how to perform a multiport calibration of the analyzer.

// **Reset** the analyzer
*RST
//
// **Set cal kit** as active kit for N50
:SENSE:CORRECTION:CKIT:N50:SELECT 'N 50 Ohm Ideal Kit'
//
// **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 in directory
// C:\Rohde&Schwarz\Nwa\Calibration\Data
// the file name in the 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'
```

# Modeling a Max Hold Function

The following example shows you how to emulate a max hold function.

```
// Reset the analyzer
*RST
:DISPlay:WINDow1:TITLe:DATA 'Max Hold Function Emulation'

// 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'
// .....
// Continous sweep mode
:INITIATE:CONTINUOUS ON
//
```

# Retrieving the Results of Previous Sweeps

The command `CALCulate<Ch>:DATA:NSWeep? SDATa, <Trace_Hist_Count>` retrieves 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

// State that INITIATE<Ch>:IMMEDIATE will start a single sweep for the
// referenced channel <Ch> only, not for all channels
:INITIATE:IMMEDIATE:SCOPE SINGLE

// Select single sweep mode and measure a single sweep group for channels no. 1 and 2
:INITIATE:IMMEDIATE:SCOPE SINGLE
:INITIATE1:IMMEDIATE; *WAI
:INITIATE2:IMMEDIATE; *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 channel 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
```

# Contents of Chapter 8


**8 Maintenance, Error Messages, Hardware Interfaces ..... 8.1**

**Maintenance ..... 8.1**

Storing and Packing ..... 8.1

**Error Messages ..... 8.2**

Global Errors ..... 8.2

Errors during the Measurement ..... 8.3

**Front Panel Connectors ..... 8.4**

Test Ports ..... 8.4

USB Connector ..... 8.4

Ground Connector ..... 8.5

Front Panel Jumpers ..... 8.5

**Rear Panel Connectors ..... 8.7**

AUX 8.7

LAN1 / LAN 2 ..... 8.7

USB 8.7

10 MHz REF ..... 8.8

DC MEAS ..... 8.8

PORT BIAS ..... 8.9

MONITOR ..... 8.9

CASCADE ..... 8.10

USER CONTROL ..... 8.10

EXT TRIGGER ..... 8.11

GPIB Bus Interface ..... 8.11

Instrument Messages ..... 8.14

RSIB Interface Functions ..... 8.14

Description of Interface Functions ..... 8.16

# 8 Maintenance, Error Messages, Hardware Interfaces

This chapter contains maintenance instructions and provides a list of hardware error messages and a detailed description of the hardware interfaces and connectors of the instrument. For a graphical overview of the front panel and rear panel connectors and their use refer to chapter *Preparing for Use:*

## Maintenance

The network analyzer does not require any special maintenance. Make sure that the air vents are not obstructed. The outside of the instrument is suitably cleaned using a soft, line-free dust cloth.

Caution!

Never use solvents such as thinners, acetone and similar things, as they may damage the front panel labeling or plastic parts.

Please follow the instructions in the service manual when exchanging modules or ordering spares. The order no. for spare parts can be found in the service manual. The service manual includes further information particularly on troubleshooting, repair, exchange of modules (including AC Fuse Board exchange, adjustment of the OCXO oscillator) and calibration.

The address of our support center and a list of useful contact addresses can be found on the *Contact* page.

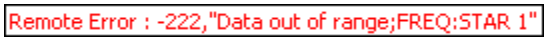
### Storing and Packing

The network analyzer can be stored at the temperature range quoted in the data sheet. When it is stored for a longer period of time the instrument should be protected against dust.

The original packing should be used, particularly the protective caps at the front and rear, when the instrument is to be transported or dispatched. If the original packing is no longer available, use a sturdy cardboard box of suitable size and carefully wrap the instrument to protect it against mechanical damage.

# Error Messages

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 three categories:

- Remote errors (SCPI errors) may 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.
- 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 generally 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 in order to avoid damage. Hardware errors with possible causes and remedies are listed below.

**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 *(Info – Error Log)*.
- Global errors can occur any time while the analyzer is operating. The analyzer is checked periodically for global errors.
- Measurement errors are due to inadmissible hardware settings and states during the measurement process.

## Global Errors

Global errors can occur any time while the analyzer is operating. The analyzer is checked periodically for global errors. Many global errors also cause an entry in the status reporting system.

| Error | Description | Remedy | Bit no.*) |
|---|---|---|---|
| Overtemperature | 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 | The analyzer detects an excess input level at a particular port. If this condition persists, all internal and external generators are switched off (Channel – Power Bandwidth Average – RF Off). | Reduce RF input level at the port. Check amplifiers in the external test setup. | 3 |
| Reference frequency lock failure | The 128 MHz TCXO is synchronized to reference frequency (OCXO, external 10 MHz reference) but the control voltage out of tolerance. | For external reference: check frequency and level of the supplied reference signal. | – |
| Converter clock frequency lock failure | The control voltage for the 80 MHz oscillators for the AD converter clock is out of tolerance. | – | 10 |

| Error | Description | Remedy | Bit no.*) |
|---|---|---|---|
| Oven cold | With option R&S ZVAB-B4, oven quartz: The oven temperature is too low. | Wait until the oven has been heated up | 8 |

*) The following bits in the STATus:QUEStionable:INTegrity:HARDware register are set when the error occurs.

## Errors during the 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 excess level at a particular port. An attenuation for the corresponding generator is activated. | Check signal path for the wave $a_i$, especially check external components. | – |
| 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. | – |
| Problem concerning external power meter Pmet<i> | An external power meter has been configured, however, it cannot be controlled or provides error messages. If several power meters cause problems, the lowest number is indicated. | Check whether the power meter is properly connected and switched on. Check the GPIB address; exclude address conflicts when using several external power meters. | – |
| 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. | – |
| DC overload | The input voltages at the DC input connectors on the rear panel are too high. | Reduce input voltage. | – |
| Unleveled output power at port <i> | The level control is unsettled or unstable, possibly due to an external disturbing signal. | Change generator level at the port; check external components. | 2 |

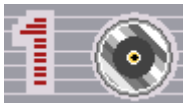
*) The following bits in the STATus:QUEStionable:INTegrity:HARDware register are set when the error occurs.

# Front Panel Connectors

## Test Ports

N-connectors, numbered 1 to 4 (for four-port instruments). 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.
- With 2, 3 or 4 test ports, it is possible to perform full two-port, 3-port or 4-port measurements; see S-Parameters section in Chapter 3.



Caution!

The maximum input levels at all test ports according to the front panel labeling or the data sheet must not be exceeded.

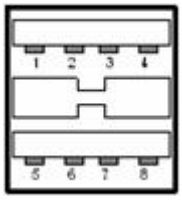
In addition, the maximum input voltages of the other input connectors at the front and rear panel must not be exceeded.

The three LEDs above each test port indicate the connector state:

- The **amber** LED is on while the connector is used as a source port.
- The **green** LED is on while the connector is used as a bidirectional (source and receive) port.
- The **blue** LED is on while the connector is used as a receive port.

## USB Connector

Double Universal Serial Bus connector of type A (master USB), used to connect e.g a keyboard, mouse or other pointing devices, the Calibration Unit (accessory R&S ZV-Z51), a printer or an external storage device (USB stick, CD-ROM drive etc.).



*The length of passive connecting USB cables should not exceed 1 m. The maximum current per USB port is 500 mA.*

## Ground Connector

Connector providing the ground of the analyzer's supply voltage.

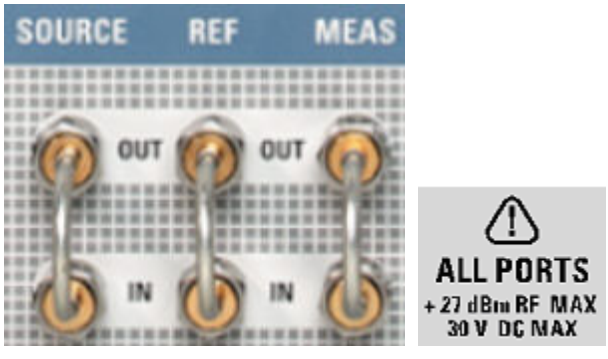
Caution!

Electrostatic discharge (ESD) may cause damage to the electronic components of the DUT and the analyzer. Use the wrist strap and cord supplied with the instrument to connect yourself to the GND connector.

## Front Panel Jumpers

3 pairs of SMA connectors for each test port. The front panel jumpers are available as option R&S ZVA<n>-B16, *Direct Generator/Receiver Access.* For detailed ordering information and application examples refer to the data sheet.

The jumpers give direct access to various RF input and output signals. They can be used to insert external components (e.g. external signal separating devices, power amplifiers, etc.) into the signal path in order to develop custom measurements, e.g. to test high power devices and extend the dynamic range. If no external components are connected, each pair of OUT/IN jumper connectors is through-connected internally.




- The SOURCE OUT signal comes from the internal RF signal source. The SOURCE IN signal goes to the test port. A power amplifier can be inserted between SOURCE OUT and SOURCE IN in order to boost the test port power.
- The REF OUT signal comes from the coupler and provides the reference signal. The REF IN signal goes to the receiver input for the reference signal.
- The MEAS OUT signal comes from the coupler and provides the received (measured) signal. The MEAS IN signal goes to the receiver input for the measured signal.

Attention!

The maximum RF input levels at all front panel jumpers inputs according to the front panel labeling or the data sheet must not be exceeded.

In addition, it is important that the signal fed in at the jumper input connectors contains no DC offset, as this may impair the measurements and even cause damage to the instrument.

# Rear Panel Connectors

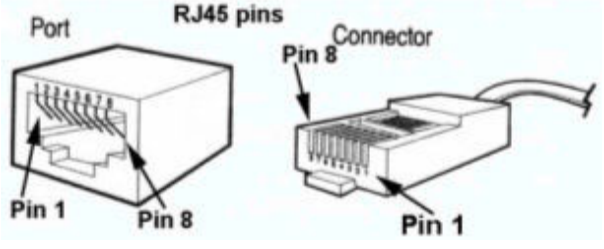
## AUX

Coaxial bidirectional auxiliary connector that can be wired as needed. The AUX connector is not fitted on standard instruments.

| Pin No. | Name | Input (I) or Output (O) or Bidirectional (B) | Voltage Range | Function |
|---|---|---|---|---|
| Center conductor | AUX | I or O or B | See data sheet | Can be wired as needed |
| Outer conductor | GNDA | – | 0 V | Analog ground |

## LAN1 / LAN 2

8-pin connector RJ-45 used to connect the analyzer to a Local Area Network (LAN). The pin assignment of the RF-45 connector supports category 5 UTP/STP (Unshielded/Shielded Twisted Pair) cables.

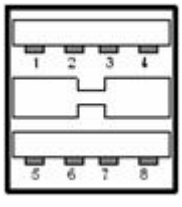



| Pin No. | Name | Input (I) or Output (O) | Voltage Range |
|---|---|---|---|
| 1 | INPUT RECEIVE DATA (+) | I | +2.5 V |
| 2 | INPUT RECEIVE DATA (–) | I | –2.5 V |
| 3 | OUTPUT TRANSMIT DATA (+) | O | +2.5 V |
| 6 | OUTPUT TRANSMIT DATA (–) | O | –2.5 V |

The other pins are not used.

## USB

Double Universal Serial Bus connector of type A (master USB), used to connect e.g a keyboard, mouse or other pointing devices, a printer or an external storage device (USB stick, CD-ROM drive etc.).



*The length of passive connecting USB cables should not exceed 1 m. The maximum current per USB port is 500 mA.*

| Pin No. | Name | Input (I) or Output (O) or Bidirectional (B) | Voltage Range | Function |
|---|---|---|---|---|
| 1 | +5.2VD | – | +5.2V | Supply voltage (rt) |
| 2 | USBD0M | B | TTL | Data D– (ws) |
| 3 | USVD0P | B | | Data D+ (gn) |
| 4 | GNDD | – | 0 V | Digital ground (sw) |
| 5 | +5.2VD | – | +5.2V | Supply voltage (rt) |
| 6 | USBD1M | B | TTL | Data D– (ws) |
| 7 | USBD1P | B | | Data D+ (gn) |
| 8 | GNDD | – | 0 V | Digital ground (sw) |

## 10 MHz REF

Coaxial BNC connector used as an input or output for the 10 MHz reference clock signal.

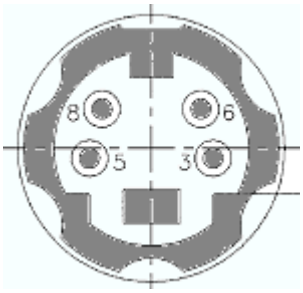
The function of the *10 MHz REF* connector depends on the *Int. Reference* or *Ext. Reference* setting in the *System* menu:

- If *Int. Reference* is active, *10 MHz REF* is used as an output connector for the 10 MHz internal reference clock signal of the analyzer.
- If *Ext. Reference* is active, *10 MHz REF* is used as an input connector for an external 10 MHz reference clock signal. The external reference signal must meet the specifications of the data sheet. The internal reference signal is synchronized to the external signal.

| Pin No. | Name | Input (I) or Output (O) or Bidirectional (B) | Voltage Range | Function |
|---|---|---|---|---|
| Center conductor | FR_REF_10_OUT/IN | B | See data sheet | Reference frequency |
| Outer conductor | GNDA | – | 0 V | Analog ground |

## DC MEAS

Two inputs for DC measurements. The connectors are of the same type but specified for different input voltage ranges, see data sheet.




| Pin No. | Name | Input (I) or Output (O) | Function |
|---|---|---|---|
| 3 | GNDA | - | Analog ground |
| 5 | GNDA | - | Analog ground |
| 6 | DCMEAS1VPOS<br>DCMEAS10VPOS | I | Positive 1 V or 10 V DC input (see data sheet) |
| 8 | DCMEAS1VNEG<br>DCMEAS10VNEG | I | Positive 1 V or 10 V DC input (see data sheet) |

## PORT BIAS

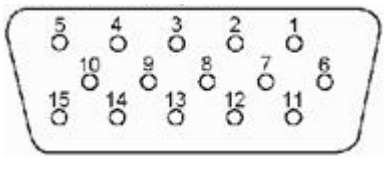
Coaxial BNC connectors used to apply an external DC voltage (bias) to the test ports. The PORT BIAS connectors are numbered 1 to 4 (for four-port analyzers).

*Each PORT BIAS input is protected by an exchangeable fuse.*

| Pin No. | Name | Input (I) or Output (O) or Bidirectional (B) | Voltage Range | Function |
|---|---|---|---|---|
| Center conductor | PORTBIAS1/2/3/4 | I | See data sheet | DC input for test port 1/2/3/4 |
| Outer conductor | GNDA | – | 0 V | Analog ground |

## MONITOR

15-pole Sub-Min-D connector used to connect and external VGA monitor.




| Pin No. | Name | Input (I) or | Voltage Range | Function |
|---|---|---|---|---|

| | | Output (O) | | |
|---|---|---|---|---|
| 1 | RED | O | 75 Ω shielded | Red signal |
| 2 | GREEN | O | 75 Ω shielded | Green signal |
| 3 | BLUE | O | 75 Ω shielded | Blue signal |
| 4 | NC | – | – | – |
| 5 | GNDA | – | 0 V | Analog ground |
| 6 | GNDA | – | 0 V | Analog ground |
| 7 | GNDA | – | 0 V | Analog ground |
| 8 | GNDA | – | 0 V | Analog ground |
| 9 | GNDA | – | 0 V | Analog ground |
| 10 | GNDA | – | 0 V | Analog ground |
| 11 | NC | – | – | – |
| 12 | DDCDAT | | | |
| 13 | | | | |
| 14 | | | | |
| 15 | | | | |

## CASCADE

8-pin RJ-45 connectors for future extensions.

## USER CONTROL

25-pole D-Sub connector used as an input and output for low-voltage (3.3 V) TTL control signals. Some of the USER CONTROL lines can be controlled via GPIB bus (see CONTrol... and OUTPut... subsystems).

| Pin No. | Name | Input (I) or Output (O) | Function |
|---|---|---|---|
| 1 | GNDD | - | Digital ground |
| 2 | EXT TRG | I | External trigger signal *) |
| 3 | GNDD | - | Digital ground |
| 4 | BUSY | O | Measurement running |
| 5 | GNDD | - | Digital ground |
| 6 | READY FOR TRIGGER | O | Measurement terminated, ready for trigger |
| 7 | GNDD | - | Digital ground |
| 8 | CHANNEL BIT 0 | O | Channel-specific bit 0;<br>see CONTrol:AUXiliary:C[:DATA] and OUTPut<Ch>:UPORt[:VALue] |
| 9 | CHANNEL BIT 1 | O | Channel-specific bit 1 |
| 10 | CHANNEL BIT 2 | O | Channel-specific bit 2 |
| 11 | CHANNEL BIT 3 | O | Channel-specific bit 3 |
| 12 | GNDD | - | Digital ground |
| 13 | PASS 1 | O | Pass/fail result of limit check *TTL Out Pass 1* |
| 14 | PASS 2 | O | Pass/fail result of limit check *TTL Out Pass 2* |
| 15 | GNDD | - | Digital ground |
| 16 | DRIVE PORT 1 | O | Test port 1 is source port (drive port) |
| 17 | DRIVE PORT 2 | O | Test port 2 is source port (drive port) |

| Pin No. | Name | Input (I) or Output (O) | Function |
|---|---|---|---|
| 18 | DRIVE PORT 3 | O | Test port 3 is source port (drive port) |
| 19 | DRIVE PORT 4 | O | Test port 4 is source port (drive port) |
| 20 | GNDD | - | Digital ground |
| 21 | EXT GEN TRG | O | Control signal for external generator |
| 22 | EXT GEN BLANK | I | Handshake signal from external generator |
| 23 | GNDD | - | Digital ground |
| 24 | FOOT SWITCH 1 | I | Control input A |
| 25 | FOOT SWITCH 1 | I | Control input B |

*) ***Note:*** *Feeding in the external trigger signal via the BNC connector EXT TRIGGER is equivalent. Both input connectors must not be used simultaneously, because this can cause malfunctions of the analyzer.*

Important note:

Use only well shielded cables or disconnect the input pins of the USER CONTROL connector in order 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 EXT TRIGGER input is used.

## EXT TRIGGER

Coaxial BNC connector used as an input for a low-voltage (3.3 V) TTL external trigger signal.

***Note:*** *Feeding in the external trigger signal at pin 2 of the D-Sub connector USER CONTROL is equivalent. Both input connectors must not be used simultaneously, because this can cause malfunctions of the analyzer.*

| Pin No. | Name | Input (I) or Output (O) or Bidirectional (B) | Voltage Range | Function |
|---|---|---|---|---|
| Center conductor | RP_EXT_TRG | I | 3.3 V TTL | External trigger signal |
| Outer conductor | GNDA | – | 0 V | Analog ground |

## GPIB Bus Interface

The standard instrument is equipped with a GPIB bus (IEC/IEEE bus) connection. The two interface connectors labeled *IEC BUS* and *IEC SYSTEM BUS* are located on the rear panel of the instrument.

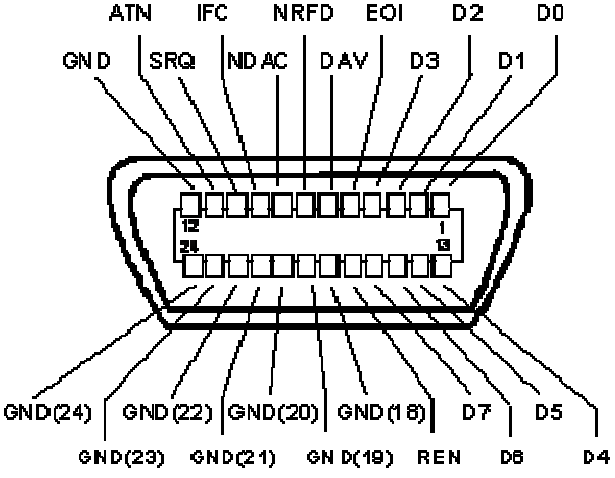
- The *IEC BUS* connector is intended for remote control of the analyzer from a controller.
- The *IEC SYSTEM BUS* can be used to control further devices from the analyzer.

*Always use a shielded cable to connect the GPIB bus interfaces.*

Characteristics of the interface

- 8-it 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
- Maximum length of the connecting cables 15 m. The length of a single connecting cable should not exceed 2 m, if many devices are used, it should not exceed 1 m.
- Wired OR if several instruments are connected in parallel

**Pin assignment**




**Bus lines**

**1. 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.

2. 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.

3.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 analyzer provides the following functions to communicate via GPIB bus:

- Interface functions
- Interface messages
- Instrument messages

## Interface Functions

Instruments which can be controlled via GPIB bus can be equipped with different interface functions. The interface function for the network analyzer 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 |

## 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 pervious 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$(24)) | Ready for serial poll. |
| SPD (Serial Poll Disable) | IBCMD (controller%, CHR$(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. |

# 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 the SCPI Reference chapter. The chapter also provides a detailed description of all messages implemented by the analyzers.

# RSIB Interface Functions

This section lists all functions of the DLL `"RSIB.DLL" or "RSIB32.DLL"` or `"librsib.so"`, which allow to produce control applications.

- Variables
- Interface Functions

## Variables ibsta, iberr, ibcntl

Same as with the National Instruments interface, successful execution of a command can be checked by means of the variables `ibsta`, `iberr` and `ibcntl`. To this end, references to the three variables are transferred to all RSIB functions. In addition, the status word `ibsta` is returned as a function value by all functions.

### Status word ibsta

All functions send back a status word that provides information on the status of the RSIB interface. The following bits are defined:

| Bit name | Bit | Hex code | Description |
|---|---|---|---|
| ERR | 15 | 8000 | This bit is set if an error occurs during a function call. If this bit is set, `iberr` contains an error code which specifies the error. |
| TIMO | 14 | 4000 | This bit is set if a timeout occurs during a function call. A timeout may occur in the following situations:<br>while waiting for an SRQ with the function `RSDLLWaitSrq()`.<br>if no acknowledgment is received for data sent to an instrument with `RSDLLibwrt()` or `RSDLLilwrt()`.<br>if no response from server to a data request with function `RSDLLibrd()` or `RSDLLilrd()`. |
| CMPL | 8 | 0100 | This bit is set if the reply of the IEC/IEEE-bus parser is completely read. If a reply of the parser is read with the function `RSDLLilrd()` and the buffer length is not sufficient, the bit is cleared. |

## Error variable iberr

If the ERR bit (8000h) is set in the status word, `iberr` contains an error code that specifies the error. The RSIB has error codes of its own independent of the National Instrument interface.

| Error | Error code | Description |
|---|---|---|
| IBERR_DEVICE_REGISTER | 1 | RSIB.DLL cannot register any new device. |
| IBERR_CONNECT | 2 | Link to the device has failed. |
| IBERR_NO_DEVICE | 3 | An interface function was called with an invalid device handle. |
| IBERR_MEM | 4 | No free memory available. |
| IBERR_TIMEOUT | 5 | Timeout has occurred. |
| IBERR_BUSY | 6 | The RSIB interface is blocked by a function not yet completed.<br>Windows is not blocked, for example, by function RSDLLibrd() if data are still to be transmitted in response to this function. In this case a new call is possible. Further calls are however rejected by RSIB.DLL with error code `IBERR_BUSY`. |
| IBERR_FILE | 7 | Error in reading from or writing to a file. |
| IBERR_SEMA | 8 | Error upon creating or assigning a semaphore (only under Unix) |

## Count variable - ibcntl

The variable ibcntl is updated with the number of bytes transmitted on every read and write function call.

## Overview of Interface Functions

The library functions are adapted to the interface functions of National Instruments for GPIB programming. The functions supported by the libraries are listed in the following table.

| Function | Description |
|---|---|
| RSDLLibfind() | Provides a handle for access to a device. |
| RSDLLibwrt() | Sends a zero-terminated string to a device. |
| RSDLLilwrt() | Sends a certain number of bytes to a device. |
| RSDLLibwrtf() | Sends the contents of a file to a device. |
| RSDLLibrd() | Reads data from a device into a string. |
| RSDLLilrd() | Reads a certain number of bytes from a device. |
| RSDLLibrdf() | Reads data from a device into a file. |
| RSDLLibtmo() | Sets timeout for RSIB functions |
| RSDLLibsre() | Switches a device into the local or remote state |
| RSDLLibloc() | Temporarily switches a device into the local state |
| RSDLLibeot() | Enables/disables the END message for write operations. |
| RSDLLibrsp() | Performs a serial poll and provides the status byte. |
| RSDLLibclr | Sends the command SDC (Device Clear) to the instrument. |
| RSDLLibonl() | Sets the device On/Offlline. |
| RSDLLTestSrq() | Checks whether a device has generated an SRQ. |
| RSDLLWaitSrq() | Waits until a device generates an SRQ. |
| RSDLLSwapBytes | Swaps the byte sequence for binary numeric display (only required for non-Intel platforms) |

# Description of Interface Functions

## RSDLLibfind()

The function provides a handle for access to the device with the name `udName`.

**VB format:** Function RSDLLibfind (ByVal udName$, ibsta%, iberr%, ibcntl&) As Integer

**C-format:** short WINAPI RSDLLibfind( char far *udName, short far *ibsta, short far *iberr, unsigned long far *ibcntl)

**C format (Unix):** short RSDLLibfind( char *udName, short *ibsta, short *iberr, unsigned long *ibcntl)

**Parameter:** `udName` IP address of device

**Example:** ud = RSDLLibfind ("89.10.38.97", ibsta, iberr, ibcntl)

The function must be called prior to all other functions of the interface.

As return value, the function provides a handle that must be indicated in all functions for access to the device. If the device with the name `udName` is not found, the handle has a negative value.

## RSDLLibwrt

This function sends data to the device with the handle `ud`.

**VB format:** Function RSDLLibwrt (ByVal ud%, ByVal Wrt$, ibsta%, iberr%, ibcntl&) As Integer

**C format:** short WINAPI RSDLLibwrt( short ud, char far *Wrt, short far *ibsta, short far *iberr, unsigned long far *ibcntl )

**C format (Unix):** short RSDLLibwrt( short ud, char *Wrt, short *ibsta, short *iberr, unsigned long *ibcntl )

**Parameter:** `ud` Device handle

`Wrt` String sent to the device.

**Example:** RSDLLibwrt(ud, "SENS:FREQ:STAR?", ibsta, iberr, ibcntl)

This function allows to send setting and query commands to the measuring instruments. Whether the data is interpreted as a complete command can be set using the function `RSDLLibeot()`.

## RSDLLilwrt

This function sends `Cnt` bytes to a device with the handle `ud`.

**VB format:** Function RSDLLilwrt (ByVal ud%, ByVal Wrt$, ByVal Cnt&, ibsta%, iberr%, ibcntl&) As Integer

**C format:** short WINAPI RSDLLilwrt( short ud, char far *Wrt, unsigned long Cnt, short far *ibsta, short far *iberr, unsigned long far *ibcntl)

**C format (Unix):** short RSDLLilwrt( short ud, char *Wrt, unsigned long Cnt, short *ibsta, short *iberr, unsigned long *ibcntl)

**Parameter:** `ud` Device handle

`Wrt` String sent to the GPIB parser.

`Cnt` Number of bytes sent to the device.

**Example:** RSDLLilwrt (ud, '.....', 100, ibsta, iberr, ibcntl)

Like `RSDLLibwrt()` this function sends data to a device. The only difference is that binary data can be sent as well. The length of the data is not determined by a zero-terminated string, but by the indication of `Cnt` bytes. If the data is to be terminated with EOS (0Ah), the EOS byte must be appended to the string.

## RSDLLibwrtf

This function sends the contents of a file `file$` to the device with the handle `ud`.

**VB format:** Function RSDLLibwrtf (ByVal ud%, ByVal file$, ibsta%, iberr%, ibcntl&) As Integer

**C format:** short WINAPI RSDLLibwrt( short ud, char far *Wrt, short far *ibsta, short far *iberr, unsigned long far *ibcntl )

**C format (Unix):** short RSDLLibwrt( short ud, char *Wrt, short *ibsta, short *iberr, unsigned long *ibcntl )

**Parameter:** `ud` Device handle

`file` File the contents of which are sent to the device.

**Example:** RSDLLibwrtf(ud, "C:\db.sav", ibsta, iberr, ibcntl)

This function allows to send setting and query commands to the measuring instruments. Whether the data is interpreted as complete command can be set using the function `RSDLLibeot()`.

## RSDLLibrd()

The function reads data from the device with the handle `ud` into the string `Rd`.

**VB format:** Function RSDLLibrd (ByVal ud%, ByVal Rd$, ibsta%, iberr%, ibcntl&) As Integer

**C format:** short WINAPI RSDLLibrd( short ud, char far *Rd, short far *ibsta, short far *iberr, unsigned long far *ibcntl )

**C format (Unix):** short RSDLLibrd( short ud, char *Rd, short *ibsta, short *iberr, unsigned long *ibcntl )

**Parameter:** `ud` Device handle

`Rd` String into which the read data is copied.

**Example:** RSDLLibrd (ud, Rd, ibsta, iberr, ibcntl)

This function fetches the responses of the GPIB parser to a query.

In the case of Visual Basic programming, a string of sufficient length must be generated before. This can be done during the definition of the string or using the command `Space$()`.

Generation of a string of the length 100: `- Dim Rd as String * 100`

- Dim Rd as String

Rd = Space$(100)

## RSDLLilrd

This function reads `Cnt` bytes from the device with the handle `ud`.

**VB format:** Function RSDLLilrd (ByVal ud%, ByVal Rd$, ByVal Cnt&, ibsta%, iberr%, ibcntl&) As Integer

**C format:** short WINAPI RSDLLilrd( short ud, char far *Rd, unsigned long Cnt, short far *ibsta, short far *iberr, unsigned long far *ibcntl )

**C format (Unix):** short RSDLLilrd( short ud, char *Rd, unsigned long Cnt, short *ibsta, short *iberr, unsigned long *ibcntl )

**Parameter:** `ud` Device handle

`cnt` Maximum number of bytes copied from the DLL into the target string `Rd`.

**Example:** RSDLLilrd (ud, RD, 100, ibsta, iberr, ibcntl)

Like the function `RSDLLibrd()`, this function reads data from a device. The only difference is that in this case the maximum number of bytes to be copied into the target string `Rd` can be indicated by means of `Cnt`. This function prevents writing beyond the end of the string.

## RSDLLibrdf()

Reads data from the device with the handle `ud` into the file `file`.

**VB format:** Function RSDLLibrdf (ByVal ud%, ByVal file$, ibsta%, iberr%, ibcntl&) As Integer

**C format:** short WINAPI RSDLLibrd( short ud, char far *file, short far *ibsta, short far *iberr, unsigned long far *ibcntl )

**C format (Unix):** short RSDLLibrd( short ud, char *file, short *ibsta, short *iberr, unsigned long *ibcntl )

**Parameter:** `ud` Device handle

`file` File into which the read data is written.

**Example:** RSDLLibrdf (ud, "c:\db.sav", ibsta, iberr, ibcntl)

The file name may as well include a drive or path specification.

## RSDLLibtmo

This function defines the timeout for a device. The default value for the timeout is set to 5 seconds.

**VB format:** Function RSDLLibtmo (ByVal ud%, ByVal tmo%, ibsta%, iberr%, ibcntl&) As Integer

**C format:** short WINAPI RSDLLibtmo( short ud, short tmo, short far *ibsta, short far *iberr, unsigned long far *ibcntl )

**C format (Unix):** short RSDLLibtmo( short ud, short tmo, short *ibsta, short *iberr, unsigned long *ibcntl )

**Parameter:** `ud` Device handle

`tmo` Timeout in seconds

**Example:** RSDLLibtmo (ud, 10, ibsta, iberr, ibcntl)

## RSDLLibsre

This function sets the device to the 'LOCAL' or 'REMOTE' state.

**VB format:** Function RSDLLibsre (ByVal ud%, ByVal v%, ibsta%, iberr%, ibcntl&) As Integer

**C format:** short WINAPI RSDLLibsre( short ud, short v, short far *ibsta, short far *iberr, unsigned long far *ibcntl)

**C format (Unix):** short RSDLLibsre( short ud, short v, short *ibsta, short *iberr, unsigned long *ibcntl)

**Parameter:** `ud` Device handle

`v` State of device

0 - local

1 - remote

**Example:** RSDLLibsre (ud, 0, ibsta, iberr, ibcntl)

## RSDLLibloc

This function temporarily switches the device to the 'LOCAL' state.

**VB format:** Function RSDLLibloc (ByVal ud%, ibsta%, iberr%, ibcntl&) As Integer

**C format:** short WINAPI RSDLLibloc( short ud, short far *ibsta, short far *iberr, unsigned long far *ibcntl)

**C format (Unix):** short RSDLLibloc( short ud, short *ibsta, short *iberr, unsigned long *ibcntl)

**Parameter:** `ud` Device handle

**Example:** RSDLLibloc (ud, ibsta, iberr, ibcntl)

After switchover to LOCAL state, the instrument can be manually operated via the front panel. On the

next access to the instrument by means of one of the functions of the library the instrument is switched again to the REMOTE state.

## RSDLLibeot

This function enables or disables the END message after write operations.

**VB format:** Function RSDLLibeot (ByVal ud%, ByVal v%, ibsta%, iberr%, ibcntl&) As Integer

**C format:** short WINAPI RSDLLibsre( short ud, short v, short far *ibsta, short far *iberr, unsigned long far *ibcntl)

**C format (Unix):** short RSDLLibsre( short ud, short v, short *ibsta, short *iberr, unsigned long *ibcntl)

**Parameter:** ud Device handle.

v 0 - no END message

1 – send END message

**Example:** RSDLLibeot (ud, 1, ibsta, iberr, ibcntl)

If the END message is disabled, the data of a command can be sent with several successive calls of write functions. The END message must be enabled again before sending the last data block.

## RSDLLibrsp

This function performs a serial poll and provides the status byte of the device.

**VB format:** Function RSDLLibrsp(ByVal ud%, spr%, ibsta%, iberr%, ibcntl&) As Integer

**C format:** short WINAPI RSDLLibrsp( short ud, char far* spr, short far *ibsta, short far *iberr, unsigned long far *ibcntl)

**C format (Unix):** short RSDLLibrsp( short ud, char *spr, short *ibsta, short *iberr, unsigned long *ibcntl)

**Parameter:** ud Device handle

spr Pointer to status byte

**Example:** RSDLLibrsp(ud, spr, ibsta, iberr, ibcntl)

## RSDLLibclr

Sends the command SDC (Device Clear) to the instrument.

**VB format:** Function RSDLLibclr(ByVal ud%, spr%, ibsta%, iberr%, ibcntl&) As Integer

**C format:** short WINAPI RSDLLibclr( short ud, short far *ibsta, short far *iberr, unsigned long far *ibcntl)

**C format (Unix):** short RSDLLibclr( short ud, short *ibsta, short *iberr, unsigned long *ibcntl)

**Parameter:** ud Device handle

**Example:** RSDLLibclr(ud, ibsta, iberr, ibcntl)

## RSDLLibonl

This function switches the device to 'online' or 'offline' mode. When it is switched to ‘offline’ mode, the interface is released and the device handle becomes invalid. By calling RSDLLibfind again, the communication is set up again.

**VB format:** Function RSDLLibonl (ByVal ud%, ByVal v%, ibsta%, iberr%, ibcntl&) As Integer

**C format:** short WINAPI RSDLLibonl( short ud, short v, short far *ibsta, short far *iberr, unsigned long far *ibcntl)

**C format:** short RSDLLibonl( short ud, short v, short *ibsta, short *iberr, unsigned long *ibcntl)

**Parameter:** ud Device handle

v Device state

0 - local

1 - remote

**Example:** RSDLLibonl(ud, 0, ibsta, iberr, ibcntl)

## RSDLLTestSRQ

This function checks the status of the SRQ bit.

**VB format:** Function RSDLLTestSrq (ByVal ud%, Result%, ibsta%, iberr%, ibcntl&) As Integer

**C format:** short WINAPI RSDLLTestSrq( short ud, short far *result, short far *ibsta, short far *iberr, unsigned long far *ibcntl)

**C format (Unix):** short RSDLLTestSrq( short ud, short *result, short *ibsta, short *iberr, unsigned long *ibcntl)

**Parameter:** ud Device handle

`result` Reference to an integer value in which the library returns the status of the SRQ bit.

0 - no SRQ

1 - SRQ active, device requests service

**Example:** RSDLLTestSrq (ud, result%, ibsta, iberr, ibcntl)

This function corresponds to the function `RSDLLWaitSrq`. The only difference is that `RSDLLTestSRQ` immediately returns the current status of the SRQ bit, whereas `RSDLLWaitSrq` waits for an SRQ to occur.

## RSDLLWaitSrq

This function waits until the device triggers an SRQ with the handle `ud`.

**VB format:** Function RSDLLWaitSrq (ByVal ud%, Result%, ibsta%, iberr%, ibcntl&) As Integer

**C format:** short WINAPI RSDLLWaitSrq( short ud, short far *result, short far *ibsta, short far *iberr, unsigned long far *ibcntl)

**C format (Unix):** short RSDLLWaitSrq( short ud, short *result, short *ibsta, short *iberr, unsigned long *ibcntl)

**Parameter:** ud Device handle

`result` Reference to an integer value in which the library returns the status of the SRQ bit.

0 - No SRQ has occurred during the timeout

1 - SRQ has occurred during the timeout

**Beispiel:** RSDLLWaitSrq( ud, result, ibsta, iberr, ibcntl );

The function waits until one of the following two events occurs.

The measuring instrument triggers an SRQ

No SRQ occurs during the timeout defined with `RSDLLibtmo()`

## RSDLLSwapBytes

This function changes the display of binary numbers on non-Intel platforms.

**VB format:** Not provided at present since it is required only on non-Intel platforms.

**C format:** void WINAPI RSDLLSwapBytes( void far *pArray, const long size, const long count)

**C format (Unix):** void RSDLLSwapBytes( void *pArray, const long size, const long count)

**Parameter:** `pArray` Array in which modifications are made

`size` Size of a single element in `pArray`

`count` Number of elements in `pArray`

**Example:** RSDLLSwapBytes( Buffer, sizeof(float), ibcntl/sizeof(float))

This function swaps the display of various elements from *Big Endian* to *Little Endian* and vice versa. It is expected that a coherent storage area of elements of the same file type (`size` byte) is transferred to `pArray`. This function has no effect on Intel platforms.

Different types of processor architecture store data in different byte sequences. For example, Intel processors store data in the reverse order of Motorola processors. Comparison of byte sequences:

| Byte sequence | Use in | Display in memory | Description |
|---|---|---|---|
| Big Endian | Motorola processors, network standard | Most significant byte at least significant address | The *most significant* byte is at the left end of the word. |
| Little Endian | Intel processors | Least significant byte at least significant address | The *most significant* byte is at the right end of the word. |

# Contents of Chapter 9

**9 Measurement Examples ..... 9.1**

**Simple Measurement Tasks ..... 9.1**

Screen Control ..... 9.1

Display Configuration ..... 9.2

Data Transfer ..... 9.2

Setting up a Sweep ..... 9.3

**Administrative Tasks ..... 9.5**

Transferring Options ..... 9.5

**Optimization ..... 9.5**

Optimizing the Measurement Speed ..... 9.5

**Calibration ..... 9.6**

**Trace Evaluation ..... 9.9**

Trace Statistics ..... 9.9

Time Domain Measurements ..... 9.10

**Port Conversion ..... 9.10**

Balanced Ports ..... 9.10

Network (De-)embedding ..... 9.12

True Differential Mode ..... 9.14

**Harmonics Measurements ..... 9.17**

**Pulse Profile ..... 9.18**

Pulsed Measurements ..... 9.18

# 9 Measurement Examples

This chapter provides examples for typical measurement tasks to be performed on the network analyzer:

To make yourself familiar with the instrument, use the simple measurement examples outlined in the Getting Started chapter. For a systematic explanation of all menus, functions and parameters and background information refer to the Manual Control reference.

## Simple Measurement Tasks

### Screen Control

Control of the analyzer's display elements is based on the standard Windows™ functionality. The following examples show you how to make efficient use of the functions.

#### To move an active setup window within the available space ...

1. Make sure that the window is not maximized (If necessary, click *Window* – *Cascade* to display all windows with their title bar).
2. Click the icon in the title bar to open the setup control menu.
3. Click *Move.*

After the pointer changes to the four-headed arrow:

4. Press one of the direction keys (left, right, up, or down arrow key) to move the entire window without changing its size.
5. Press *ENTER* or left-click the window when it is at the position you want.

#### To size the active setup window ...

1. Make sure that the window is not maximized (If necessary, click *Window* – *Cascade* to display all windows with their title bar).
2. Click the icon in the title bar to open the setup control menu.
3. Click *Size.*

After the pointer changes to the four-headed arrow:

4. Press one of the direction keys (left, right, up, or down arrow key) to move the pointer to the border you want to move.
5. Press a direction key to move the border.

Press *ENTER* or left-click the window when it is the size you want.

# Display Configuration

The analyzer provides various tools that you can use to customize the diagram areas and control elements. The following examples show how to change the display colors.

### To select or define a color scheme for the diagrams and their display elements...

1. Click Display — Display Config — Color Scheme... to open the Color Scheme dialog.
2. Select the *Predefined Scheme* that is most appropriate for your needs.
3. Click *Define User Scheme...* to open the *Define User Color Scheme* dialog.
4. Select a display element from the *Element* drop-down list.
5. Click the *Color* field to open a standard *Color* dialog and assign the color.

You can repeat the last 2 steps for as many elements as you like. For traces, you can also select a *Trace Style* and *Trace Width.* Your settings apply to all setups, they are permanent even after you close the network analyzer application.

6. If you wish to use several color schemes in parallel, click *Save* to save your current color scheme to a file.
7. If you wish to restore the original *Predefined Scheme,* click *System — System Config... — Resets — Reset Colors.*

### To change the marker colors...

When you create markers, their color is the same as the color of the associated trace. You can change the marker colors using the *Color Scheme* dialog.

1. Click Display — Display Config — Color Scheme...
2. Click *Define User Scheme...* to open the *Define User Color Scheme* dialog.
3. Select *Same Color for all Markers* from the *Element* drop-down list.
4. Click the *Color* field to open a standard *Color* dialog and assign the color.
5. Click *Use same Color for all Markers.*

All markers are displayed with the same color, irrespective of the trace colors.

# Data Transfer

The following examples show you how to perform basic tasks related to data transfer from and to the analyzer.

### To copy the active setup ...

1. Insert a floppy disk into the disk drive.
2. Click *File — Save* to open the *Save As* dialog.
3. Open the *Save in:* drop-down list and select disk drive A (labeled 3 1/2 Floppy (A)) as file location.
4. Select a *File name* and click *Save* to store the data to the disk and close the *Save As* dialog.

The default extension for setup files (*.nwa) is appended automatically.

### To load a setup stored on a floppy disk...

5. Insert the floppy disk into the disk drive.
6. Click *File — Open* to open the *Open* dialog.
7. Open the *Look in:* drop-down list and select disk drive A (labeled 3 1/2 Floppy (A)) as file location.

The setup files are displayed in the center of the dialog.

1. Select a file and click *Open* to load the setup as the active setup.

## Setting up a Sweep

Configurations of the different sweep types require different procedures. The following examples list the essential settings for various sweep types.

### To Set up a Lin. Frequency Sweep...

1. Press the *Sweep* key in the *Channel* keypad or click *Channel - Sweep* to access the Sweep submenu.
2. Click *Sweep Type – Lin. Frequency.*
3. Click *Channel – Stimulus* to access the submenu defining the sweep range.
4. To define a sweep range confined by a start and a stop frequency, use *Start* and *Stop.* Alternatively, to define a sweep range of definite width around a known center frequency (e.g. an expected peak), use *Center* and *Span.*
5. To vary the number of measurement points (e.g. to improve the resolution of the sweep or reduce the sweep time), click *Channel – Sweep – Number of Points.*

If the active diagram is a Cartesian diagram, the x-axis is linearly scaled and labeled with the *Start* and *Stop* frequencies.

### To Set up a Log. Frequency Sweep...

1. Press the *Sweep* key in the *Channel* keypad or click *Channel - Sweep* to access the *Sweep* submenu.
2. Click *Sweep Type - Log. Frequency.*
3. Click *Channel - Stimulus* to access the submenu defining the sweep range.
4. To define a sweep range confined by a start and a stop frequency, use *Start* and *Stop.* Alternatively, to define a sweep range of definite width around a known center frequency (e.g. an expected peak), use *Center* and *Span.*
5. To vary the number of measurement points (e.g. to improve the resolution of the sweep or reduce the sweep time), click *Channel - Sweep - Number of Points.*

If the active diagram is a Cartesian diagram, the x-axis is logarithmically scaled and labeled with the *Start* and *Stop* frequencies.

## To Set up a Segmented Frequency Sweep...

1. Press the *Sweep* key in the *Channel* keypad or click *Channel - Sweep* to access the Sweep submenu.
2. Click *Sweep Type - Define Segments* to call up the dialog defining the individual sweep segments. Proceed as described in Define Segments to customize your sweep range. Make sure to define at least 2 distinct sweep frequencies.
3. Use the controls in the *Individual Settings* panel to define the internal generator power, measurement (IF) bandwidth and other measurement settings for each sweep segment.
4. Close the *Define Segments* dialog.
5. Click *Sweep Type - Segmented Frequency.*

If the active diagram is a Cartesian diagram, the x-axis is linearly scaled and ranges from the lowest to the highest frequency in all segments.

## To Set up a Power Sweep...

1. Press the *Sweep* key in the *Channel* keypad or click *Channel - Sweep* to access the *Sweep* submenu.
2. Click *Sweep Type - Power.*
3. In the *CW Frequency* entry bar opened, enter the fixed stimulus frequency for the power sweep.
4. Click *Channel - Stimulus* to access the submenu defining the sweep range.
5. To define the start and stop power, use *Start* and *Stop.*
6. To vary the number of measurement points (e.g. to improve the resolution of the sweep or reduce the sweep time), click *Channel - Sweep - Number of Points.*

If the active diagram is a Cartesian diagram, the x-axis is linearly scaled (in dB) and labeled with the *Start* and *Stop* powers.

## To Set up a Time Sweep...

1. Press the *Sweep* key in the *Channel* keypad or click *Channel - Sweep* to access the *Sweep* submenu.
2. Click *Sweep Type - Time.*
3. In the *Time Sweep entry bar* opened, enter the fixed stimulus frequency *(CW Frequency)* and the internal generator power *(Source Power)* for the *Time* sweep.
4. Click *Channel - Stimulus* to access the submenu defining the sweep range.
5. To vary (extend) the sweep time, use *Stop.*
6. To vary the number of measurement points (e.g. to improve the resolution of the sweep or reduce the sweep time), click *Channel - Sweep - Number of Points.*

If the active diagram is a Cartesian diagram, the x-axis is linearly scaled and labeled with the start time 0 s and the *Stop* time.

## To Set up a CW Mode Sweep...

1. Press the *Sweep* key in the *Channel* keypad or click *Channel - Sweep* to access the *Sweep* submenu.

2. Click *Sweep Type - CW Mode.*
3. In the *CW Mode* entry bar opened, enter the fixed stimulus frequency *(CW Frequency)* and the internal generator power *(Source Power)* for the sweep.
4. Click *Channel - Stimulus* to access the submenu defining the sweep range.
5. To vary the number of measurement points (implying a longer sweep time), use *Stop.*

If the active diagram is a Cartesian diagram, the x-axis is linearly scaled and labeled with the current numbers of the trigger events (measurement points).

# Administrative Tasks

## Transferring Options

Software options can be manually deinstalled on a ZVA analyzer in order to be transferred to another instrument. Proceed as follows:

1. Contact your Rohde & Schwarz representative in order to obtain a deinstallation code.
2. Enter the deinstallation code in the *Option Key Entry* tab of the *System Config* menu and press *Install.*

The analyzer displays a notice box with a response code.

3. Note the response code and send it to your Rohde & Schwarz representative in order to obtain a new option key. You can repeat step 2 and retrieve the response code again, should you inadvertently close the notice box.
4. With the new option key, install the software option on another ZVA network analyzer.

# Optimization

## Optimizing the Measurement Speed

Minimizing the measurement time per sweep contributes to a high measurement throughput. The following parameters have an influence on the sweep time:

### To find the best set of sweep points...

Limit the measurement to the stimulus range that you need for your device:

1. Click *Channel - Stimulus* to access the submenu defining the sweep range.
2. To define a sweep range confined by a start and a stop frequency, use *Start* and *Stop*. Alternatively, to define a sweep range of definite width around a known center frequency (e.g. an expected peak), use *Center* and *Span*. For power and time sweeps, proceed as described in Setting up a Sweep.
3. Alternatively, use the marker functions to define the sweep range.

Use the minimum number of points that will produce an acceptable resolution:

1. Click *Channel – Sweep – Number of Points* and reduce the number of sweep points.

Use the segmented sweep to focus on different subranges of the sweep, using optimized channel settings for each subrange:

1. Set up a segmented frequency sweep as described in Segmented Frequency.
2. Optimize the number of points and the noise reduction settings for each individual sweep segment.

The sweep time increases with the *Frequency Step Size.*

### To obtain the fastest measurement sequence...

Avoid delay times between the different partial measurements:

1. Click Channel – Sweep – Sweep Time and check Auto.

The analyzer determines the minimum sweep time according to the current channel settings. The sweep is performed at maximum speed without delay.

Avoid delay times between the end of the measurement sequence and the next trigger event:

2. If you don't need a special trigger mode, click *Channel – Sweep – Trigger* and make sure that *Free Run* is selected.

Avoid delay times between adjacent frequency points.

3. Click *Channel – Mode* and select *Alternating Sweeps* if the settling time between adjacent frequency points is larger than the settling time between the partial measurements.

### To find the appropriate noise reduction settings...

Use the fastest IF filter that will produce acceptable dynamic range:

1. Press the *POWER BW AVG* key in the *Channel* keypad or click *Channel – Power Bandwidth Average* to access the corresponding submenu.
2. Select the widest bandwidth compatible with your measurement task.
3. If possible, click *Fine Adjust* to further increase the bandwidth. Avoid selecting *High Selectivity* unless it is required for your measurement.

Reduce the number of repeated measurements made to obtain an averaged result. Averaging does not slow down the measurement, however, several consecutive sweeps must be performed until an average result is available.

1. If averaging is not necessary, open the *Channel – Power Bandwidth Average* menu and make sure that *Average On* is not active.
2. Alternatively, click *Average Factor* and reduce the factor to a minimum.

# Calibration

Calibration is the process of eliminating systematic, reproducible errors from the measurement results (system error correction). A measurement channel can be calibrated either manually or automatically using the Calibration Unit (accessory R&S ZV-Z51/-Z52 or R&S ZV-Z58):

## Select and perform a manual calibration

In the following example a 2-port TOSM calibration is performed manually using the Calibration Wizard. Other calibrations can be performed in an analogous way.

1. Perform all channel settings for your measurement.
2. Click *Channel – Calibration – Start Cal – Two Port P1 P2 – Manual...* to open the calibration wizard for the desired calibration type.
3. In the *Select Physical Port Connector(s)* dialog, select the connector type that you use at port 1 and 2 (both connector types must be equal) and the calibration kit. Click *Next >* to access the *Measure Standards* dialog.
4. Connect an open standard to port 1 and click the corresponding box in the *Measured Standards* list to initiate the calibration sweep and acquire the calibration data for this standard.
5. Repeat the last step for a short and a match connected to port 1, an open, short and match connected to port 2, and for a through between both ports.
6. Click *Apply* to start the calculation of the system error correction data, apply them to the current channel and close the calibration wizard.
7. Replace the last measured standard with your DUT and perform calibrated measurements without changing the channel settings used for calibration.

You can also access manual calibration from the *Measurement Wizard.*

## Perform an automatic calibration

In the following example a 4-port TOSM calibration is performed using the calibration unit (accessory R&S ZV-Z51/-Z52 or R&S ZV-Z58). The calibration of other numbers of ports is analogous; see Performing an Automatic Calibration.

Connect the calibration unit:

1. Switch on and start-up your network analyzer and establish a USB connection to the calibration unit; see Connecting the calibration unit.
2. Connect ports 1 the unit to port 1 of the analyzer; repeat this for ports 2 to 4.

Perform the calibration:

3. Click *Channel – Calibration – Start Cal – Four Port P1 P2 P3 P4 – Calibration Unit* and wait until the calibration is completed. If a message box *All test port connectors will be changed to <Connector type>* pops up, confirm with *OK.*
4. Remove the connecting cables from the unit, connect your DUT instead and perform calibrated measurements.

You can also access automatic calibration from the *Measurement Wizard.*

## TRL calibration with 2 lines

Suppose you want to perform a TRL calibration using coaxial air lines for the sweep range between 1 GHz and 40 GHz. The through standard has length $l_{thr}$. The ratio of the stop frequency to the start frequency is 40, therefore an accurate calibration with a single line standard is not possible. However, it is possible to calibrate the entire sweep range with two lines of different (electrical) lengths $l_{long}$ and $l_{short}$; see *TRL Extensions.*

**To determine the electrical lengths of the two lines...**

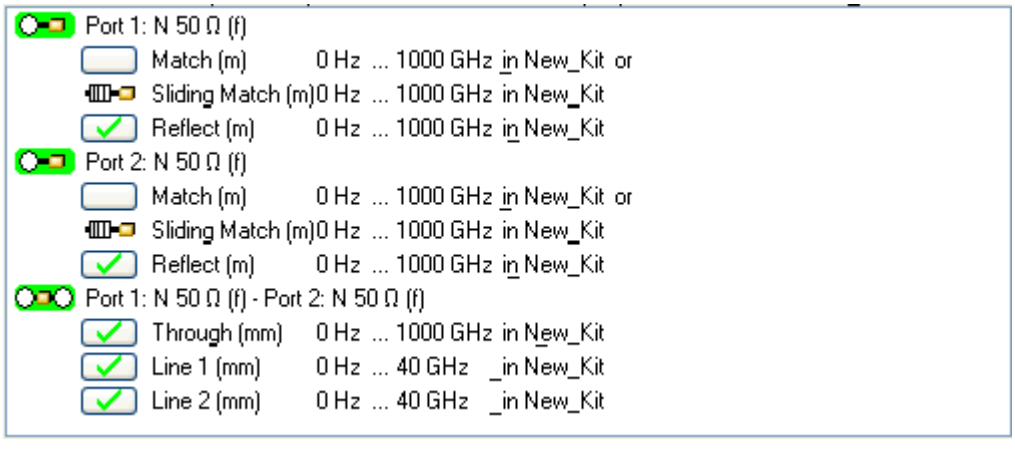
1. Select the length of the longer line such that the transmission phase of ($l_{long}$ – $l_{thr}$) at the start

frequency of 1 GHz is equal to 20 deg: $l_{long} = \lambda$ *(1 GHz) / 18 +* $l_{thr} = c_0$ */ (18 * 1 GHz) +* $l_{thr}$ = 1.666 cm + $l_{thr}$, where $c_0$ is the velocity of light. This length corresponds to a delay time of $\Delta t_{long}$ = 55.55 ps + $l_{thr}/c_0$.

2. Select the length of the shorter line such that the transmission phase of ($l_{short} - l_{thr}$) at the stop frequency of 40 GHz is equal to 160 deg: $l_{short} = 4 * \lambda$ *(40 GHz) / 9+* $l_{thr} = 4 * c_0$ */ (9 * 40 GHz)+* $l_{thr}$ = 0.333 cm+ $l_{thr}$. This length corresponds to a delay time of $\Delta t_{short}$ = 11.11 ps + $l_{thr}/c_0$.

**To perform the calibration...**

3. Make sure that the two lines are included in a calibration kit, together with the through and the reflect standard required for the TRL calibration. If necessary, click *Channel – Calibration – Cal Kits...* and use the *Calibration Kits* dialog to select a kit, add the two lines, and define their delay times as determined above. Note that the gender combinations (mm, ff or mf) of both lines must be equal.
4. Still in the *Calibration Kits* dialog, click *Export Kit...* to save your settings to a cal kit file.
5. Close the *Calibration Kits* dialog.
6. Click *Channel – Calibration – Start Cal – Two Port P1 P2 – TRL...* to open the calibration wizard (see also Manual calibration).
7. In the *Physical VNA Test Port Connector(s)* dialog, select your *Connector* type and *Calibration Kit*. Click *Next >.*
8. In the *Measured Standards* dialog opened, perform the calibration sweeps for all standards including the two displayed lines *Line 1* and *Line 2.*



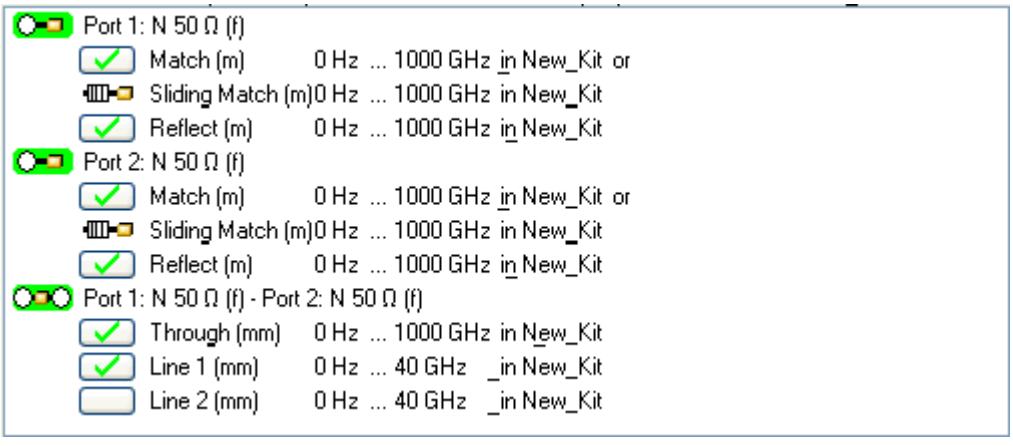

9. Click *Apply* to start the calculation of the system error correction data, apply them to the current channel and close the calibration wizard.
10. Replace the last measured standard with your DUT and perform calibrated measurements without changing the channel settings used for calibration.

## TRL calibration with low-frequency TRM extension

Suppose you want to perform a TRL calibration using coaxial air lines for the sweep range between 300 MHz and 4 GHz using a line standard with an electrical length of $l_{long}$ = 16.6666 cm and a through standard of $l_{thr}$. << $l_{long}$. TRL calibration is known to become inaccurate in the frequency range below $f_{TRM} = c_0/(18*l_{long})$ = 1 GHz; see *TRL Extensions.* It is therefore advisable to perform a combined TRL/TRM calibration using an additional match or sliding match standard.

1. Make sure that the calibration kit contains a match or sliding match standard, in addition to the line, the through and the reflect standards required for the TRL calibration. If necessary generate a cal kit file as described in section *TRL calibration with 2 lines*.

2. Click *Channel – Calibration – Start Cal – Two Port P1 P2 – TRL...* to open the calibration wizard (see also Manual calibration).
3. In the *Physical VNA Test Port Connector(s)* dialog, select your *Connector* type and *Calibration Kit*. Click *Next >.*
4. In the *Measured Standards* dialog opened, perform the calibration sweeps for the match (or sliding match) and the reflect standard at all ports. In addition, perform the calibration sweeps for the through and the line *(Line 1)* between any combination of ports.




5. Click *Apply* to start the calculation of the system error correction data, apply them to the current channel and close the calibration wizard.
6. Replace the last measured standard with your DUT and perform calibrated measurements without changing the channel settings used for calibration.

# Trace Evaluation

## Trace Statistics

After measuring a quantity across the selected sweep range, the analyzer can analyze the trace and determine various statistical results.

### To determine the x-dB compression point...

1. Press the *Sweep* key in the *Channel* keypad or click *Channel - Sweep* to access the Sweep submenu.
2. Click *Sweep Type - Power.* In the *CW Frequency* input field opened, enter the fixed stimulus frequency for the power sweep.
3. Click *Channel - Stimulus - Start* and *Channel - Stimulus - Start* to define the power sweep range.
4. Click *Trace – Measure* and select an *S-Parameter* or a *Ratio* to be measured.
5. Click *Trace – Format* and ensure that the trace format is *dB Mag*.
6. Click *Trace Funct – Trace Statistics – Compression Point* to activate the compression point evaluation.

A marker is placed to the compression point. To refine the measurement, you can vary the

*Compression Value* and the *Evaluation Range* in the *Trace Statistics* submenu.

## Time Domain Measurements

With option ZVAB-K2, *Time Domain,* you can view the measurement results as a function of time.

### To set up a time domain measurement...

1. Reset the analyzer to ensure that the following operating mode is set: *Sweep Type: Lin. Frequency, Trace - Meas.: S21, Trace - Format: dB Mag*.
2. Open the *Trace - Meas* menu and select the measured quantity to be transformed (preferably: an S-parameter or ratio).
3. Click *Trace - Trace Func - Transform - Time Domain* to transform the active trace to the time domain.

The transformed trace is displayed in the active diagram area. To refine the analysis, you can modify the number and position of the sweep points *(Channel - Sweep - Number of Points, Channel - Stimulus),* select a different transform type and window for profiling *(Trace - Trace Func - Transform - Define Transform...)* and vary the time range *(Trace - Trace Func - Transform - Time Domain Stimulus Axis...).*

### To eliminate an unwanted response from the measured trace...

1. Calculate a time domain transform as described above, Setting up a time domain measurement.
2. Click *Trace - Trace Func - Transform - Define Time Gate* and place a time gate on the transformed trace. To remove a single response use a *Notch* gate type. To everything outside a specified area, use a *Bandpass* gate type.
3. Click *Trace - Trace Func - Transform - Time Gate* to enable the time gate. If necessary, go back to the *Define Time Gate* dialog to correct your time gate settings.
4. Click *Trace - Trace Func - Transform - Frequency Domain.*

The frequency domain trace now corresponds to the modified time domain trace which is transformed back into the frequency domain.

# Port Conversion

## Balanced Ports

The *Balanced and Measured Ports* dialog comprises the functions for balanced port definition. The following examples show how to define and use balanced ports.

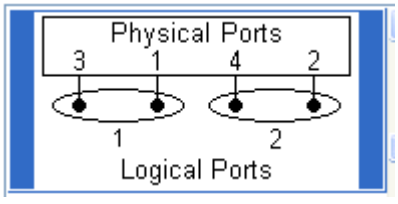
### To perform a balanced measurement...

The following example shows how to perform S-parameter measurements on a 2-port DUT with balanced input and balanced output using a 4-port network analyzer. You can either use a predefined

port configuration with fixed port assignments or define your own port configuration.

**To use a predefined port configuration,**

1. Click Trace – Measure – More S-Params... – Balanced and Measured Ports....
2. In the *Predefined Configs* tab of the *Balanced and Measured Ports* dialog opened, select the port configuration that is suitable for your DUT.



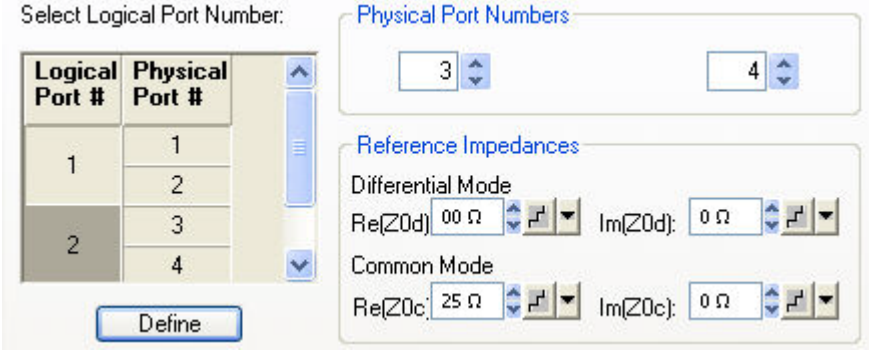

3. Open the *Def Balanced Port* tab and assign the appropriate differential and common mode reference impedances to both logical ports.
4. Connect your DUT to ports 1 to 4 of the network analyzer as shown in the figure above.
5. Click *OK* to close the *Balanced and Measured Ports* dialog.
6. Back in the *More S-Parameters* dialog, select the balanced S-parameter that you wish to measure.
7. Click *OK* to close the *More S-Parameters* dialog and perform the measurement.

TIP You can also access all the described settings from the *Measurement Wizard.*

**To use your own port configuration,**

1. Click Trace – Measure – More S-Params... – Balanced and Measured Ports....
2. In the *Def Balanced Port* tab of the *Balanced and Measured Ports* dialog opened, select *Logical Port # 1* and assign two *Physical Port Numbers,* e.g. 1 and 2.
3. Assign the appropriate differential and common mode reference impedances to logical port no. 1 and Click *Define*.
4. Select *Logical Port # 2* and assign the two remaining *Physical Port Numbers,* e.g. 3 and 4.
5. Assign the appropriate differential and common mode reference impedances to logical port no. 2 and Click *Define*.

The *Def Balanced Port* tab shows the balanced port configuration:




6. Connect one balanced port of your DUT to ports 1 and 2 of the network analyzer, the other balanced port of your DUT to ports 3 and 4 of the network analyzer, in accordance with the selected reference impedances.

7. Click *OK* to close the *Balanced and Measured Ports* dialog.
8. Back in the *More S-Parameters* dialog, select the balanced S-parameter that you wish to measure.
9. Click *OK* to close the *More S-Parameters* dialog and perform the measurement.

## Network (De-)embedding

The *Virtual Transform* menu provides the functions for network embedding and deembedding. The following examples show how to use the functions.

### To add a virtual transformation network to a single-ended port...

Suppose that your DUT is a 2-port DUT with single ended ports.

1. Connect the DUT to your analyzer: For a two-port measurement, connect the DUT between test ports 1 and 2 of the analyzer.
2. Establish the necessary *Channel* settings (port configuration, sweep range and type etc.) and select the measured quantities *(Trace – Measure).*
3. Click *Channel – Mode – Virtual Transform – Single Port Embedding* to open the *Single Port Embedding...* dialog.
4. In the dialog, select the *Physical Port* to which you want to add a virtual transformation network (e.g. *Physical Port 1)* and do one of the following:
5. If you wish to define the added network by an imported set of 2-port S-parameters, select the *2-Port* transformation network, click *Read Data From File,* and load the parameters from a 2-port (*.s2p) Touchstone file.
6. If you wish to define the added network by an equivalent circuit, select the circuit type and adjust the parameters R, C, and L displayed in the right half of the dialog.
7. Click *Embed DUT* and *Close* the dialog.

The traces of the active channel show the characteristics of the DUT including the virtual transformation network inserted between port 1 of the analyzer and the DUT.

You can also assign two-port networks to balanced ports; see *Combination of 2-Port and 4-Port (De-)Embedding Networks.*

### To remove a real transformation network from a single-ended port...

Suppose that your 2-port DUT has a real, known 2-port transformation network at port 1.

1. Connect the DUT (including the transformation network) between test ports 1 and 2 of the analyzer so that the transformation network is placed between port 1 of the analyzer and the DUT.
2. Establish the necessary *Channel* settings (port configuration, sweep range and type etc.) and select the measured quantities *(Trace – Measure).*
3. Click *Channel – Mode – Virtual Transform – Single Port Deembedding* to open the *Single Port Deembedding...* dialog.
4. In the dialog, select the *Physical Port* connected to the physical transformation network (e.g. *Physical Port 1)* and do one of the following:

5. If your transformation network is described by an imported set of 2-port S-parameters, select the *2-Port* transformation network, click *Read Data From File,* and load the data from a 2-port (*.s2p) Touchstone file.
6. If your transformation network can be described by an equivalent circuit, select the circuit type and adjust the parameters R, C, and L displayed in the right half of the dialog.
7. Click *Deembed DUT* and *Close* the dialog.

The traces of the active channel show the characteristics of the DUT without the effects of the removed physical transformation network.

You can also assign two-port networks to balanced ports; see *Combination of 2-Port and 4-Port (De-)Embedding Networks.*

## To add a virtual transformation network to a balanced port...

Suppose that your DUT has a single-ended and a balanced port and that you want to add a virtual 4-port network to the balanced port.

1. Connect ports 1 and 3 of a 3- or 4-port analyzer to the balanced ports of the DUT, connect port 2 to the single ended port of the DUT.
2. Click *Channel – Mode – Port Config... – Balanced and Measured Ports... Predefined Configs* and define the appropriate balanced port configuration (see also *Balanced Measurement Examples)*, combining ports 1 and 3 of the analyzer to logical port no. 1.
3. Establish the necessary *Channel* settings (sweep range and type etc.) and select the measured quantities *(Trace – Measure).*
4. Click *Channel – Mode – Virtual Transform – Balanced Port Embedding* to open the *Balanced Port Embedding...* dialog.

In the dialog, the *Logical Port* no. 1 (the only balanced port) is already selected. To add a virtual network, do one of the following:

5. If you wish to define the added network by an imported set of 4-port S-parameters, select the *4-Port* transformation network, click *Read Data From File,* and load the parameters from a 4-port (*.s4p) Touchstone file.
6. If you wish to define the added network by an equivalent circuit, select the circuit type and adjust the parameters R, C, and L displayed in the right half of the dialog.
7. If you wish to use a combination of 2-port S-parameters and lumped circuit elements, select the appropriate *Serial 2-Port..* transformation network, import the 2-port files for each of the physical ports 1 and 3, and adjust the remaining R, C, and L parameters.
8. Click *Embed DUT* and *Close* the dialog.

The traces of the active channel show the characteristics of the DUT including the virtual transformation network inserted between the logical port 1 of the analyzer and the balanced DUT port.

## To remove a real transformation network from a balanced port...

Suppose that your DUT has a single-ended and a balanced port and a real, known 4-port transformation network at the balanced port.

1. Connect the DUT (including the transformation network) to your 3- or 4-port analyzer: Connect ports 1 and 3 of the analyzer to the outer connectors of the transformation network, connect port 2 to the single ended port of the DUT.

2. Click *Channel – Mode – Port Config... – Balanced and Measured Ports... Predefined Configs* and define the appropriate balanced port configuration (see also *Balanced Measurement Examples),* combining ports 1 and 3 of the analyzer to logical port no. 1.
3. Establish the necessary *Channel* settings (sweep range and type etc.) and select the measured quantities *(Trace – Measure).*
4. Click *Channel – Mode – Virtual Transform – Balanced Port Deembedding* to open the *Balanced Port Deembedding...* dialog.

In the dialog, the *Logical Port* no. 1 (the only balanced port) is already selected. To virtually remove the physical network, do one of the following:

5. If you wish to describe the network by an imported set of 4-port S-parameters, select the *4-Port* transformation network, click *Read Data From File,* and load the parameters from a 4-port (*.s4p) Touchstone file.
6. If you wish to describe the network by an equivalent circuit, select the circuit type and adjust the parameters R, C, and L displayed in the right half of the dialog.
7. If you wish to use a combination of 2-port S-parameters and lumped circuit elements, select the appropriate *Serial 2-Port..* transformation network, import the 2-port files, and adjust the remaining R, C, and L parameters.
8. Click *Deembed DUT* and *Close* the dialog.

The traces of the active channel show the characteristics of the DUT without the effects of the removed physical transformation network.

# True Differential Mode

In *True Differential Mode,* the analyzer can generate balanced waves at arbitrary reference planes in the test setup and determine balanced results. The true differential mode also provides two additional sweep types.

## Measuring balanced wave quantities and ratios...

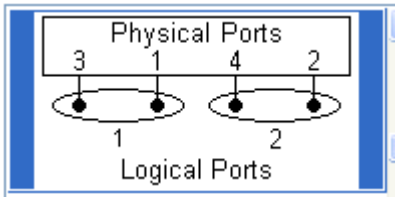
The following example shows how to activate the true differential mode in order to perform balanced measurements on a 2-port DUT with balanced input and balanced output using a 4-port network analyzer. You can use a predefined port configuration with fixed port assignments.

**To prepare the measurement,**

1. Perform a full calibration of all four analyzer ports, preferably using a calibration unit for automatic (TOSM) calibration. You can also use the calibration wizard, e.g. if you want to use one of the 7-term calibration types (TNA, TRL, ...); see *Calibrating a Measurement Channel.*
2. Preferably, perform a source power calibration at the input of your DUT.

**To obtain balanced ratios,**

1. Click Trace – Measure – Ratios – More Ratios...
2. In the *Port Configuration* panel of the *More Ratios* dialog opened, click *Balanced and Measured Ports.*
3. In the *Predefined Configs* tab of the *Balanced and Measured Ports* dialog opened, select the port configuration that is suitable for your DUT.




*The true differential mode requires logical port combinations with independent power sources. Do not use ports 1/2 or ports 3/4 to create a logical port.*

4. Still in the *Predefined Configs* tab, enable *True Differential Mode.*
5. Open the *Def Balanced Port* tab and assign the appropriate differential and common mode reference impedances to both logical ports.
6. Click *OK* to close the *Balanced and Measured Ports* dialog.
7. Connect one balanced port of your DUT to ports 1 and 3 of the network analyzer, the other balanced port of your DUT to ports 2 and 4 of the network analyzer, in accordance with the selected port combination and reference impedances.
8. Back in the *More Ratios* dialog, select the balanced ratio that you wish to measure.
9. Click *OK* to close the *More Ratios* dialog and perform the measurement.

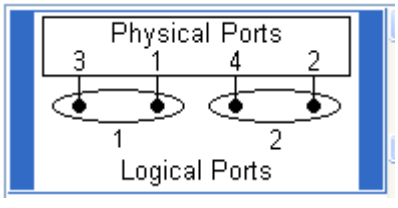
**To obtain balanced wave quantities,**

1. Calibrate your test setup, define the balanced port configuration and set the reference impedances as described above.
2. Click *Trace – Measure – Wave Quantities – More Wave Quantities...*
3. Select the balanced wave quantity that you wish to measure.
4. Click *OK* to close the *More Wave Quantities* dialog and perform the measurement.

## Measuring balanced S-parameters...

The following example shows how to activate the true differential mode in order to obtain mixed-mode S-parameters. We use a 2-port DUT with balanced input and balanced output and a 4-port network analyzer as described above.

1. Calibrate your test setup as outlined above (Measuring balanced wave quantities and ratios)
2. Click *Trace – Measure – More S-Params...*
3. In the *Port Configuration* panel of the *More S-Parameters* dialog opened, click *Balanced and Measured Ports.*
4. In the *Predefined Configs* tab of the *Balanced and Measured Ports* dialog opened, select the port configuration that is suitable for your DUT.




*The true differential mode requires logical port combinations with independent power sources. Do not use ports 1/2 or ports 3/4 to create a logical port.*

5. Still in the *Predefined Configs* tab, enable *True Differential Mode.*
6. Open the *Def Balanced Port* tab and assign the appropriate differential and common mode reference impedances to both logical ports.
7. Click *OK* to close the *Balanced and Measured Ports* dialog.
8. Connect one balanced port of your DUT to ports 1 and 3 of the network analyzer, the other balanced port of your DUT to ports 2 and 4 of the network analyzer, in accordance with the selected port combination and reference impedances.
9. Back in the *More S-Parameters* dialog, select the mixed-mode S-parameter that you wish to measure.
10. Click *OK* to close the *More S-Parameters* dialog and perform the measurement.

You can also access all the described settings from the *Measurement Wizard.*

## Performing an amplitude or phase imbalance sweep...

Amplitude or phase imbalance sweeps are special sweep types in true differential mode. The analyzer generates a balanced signal at one of its logical ports, however, the amplitude of one signal component or the relative phase of the two components is varied according to a selected sweep range.

In the following examples we use a 2-port DUT with balanced input and balanced output and a 4-port network analyzer as described above.

1. Calibrate your test setup, define the appropriate port configuration and enable true differential mode as outlined above (Measuring balanced wave quantities and ratios).

**To perform an amplitude imbalance sweep,**

2. Click *Channel – Sweep – Sweep Type: Amplitude Imbalance.*
3. In the *Amplitude Imbalance Sweep* dialog opened, select the logical Port 1 (comprising the physical ports 1 and 3) as the swept port. The second logical port will continue providing an undisturbed balanced stimulus signal.
4. Select the *CW Frequency* and the reference power (*Source Power at 0 dB)* for the sweep.
5. Close the dialog, click *Channel – Stimulus – Start* to define the start power of port 3 relative to the *Source Power at 0 dB.*
6. Click *Channel – Stimulus – Stop* to define the stop power of port 3 relative to the *Source Power at 0 dB.*
7. Click *Trace – Measure – More S-Params...* and select the measured mixed-mode S-parameter.
8. Click *OK* to close the *More S-Parameters* dialog and perform the measurement.

**To perform a phase imbalance sweep,**

2. Click Channel – Sweep – Sweep Type: Phase Imbalance.
3. In the *Phase Imbalance Sweep* dialog opened, select the logical Port 1 (comprising the physical ports 1 and 3) as the swept port. The second logical port will continue providing an undisturbed balanced stimulus signal.
4. Select the *CW Frequency* and the constant *Source Power* for the sweep.
5. Close the dialog, click *Channel – Stimulus – Start* to define the start phase difference between the physical ports 3 and 1.
6. Click *Channel – Stimulus – Stop* to define the stop phase difference between the physical ports 3 and 1.
7. Click *Trace – Measure – More S-Params...* and select the measured mixed-mode S-parameter.

8. Click *OK* to close the *More S-Parameters* dialog and perform the measurement.

# Harmonics Measurements

In the *Harmonic Distortion* menu you can measure arbitrary harmonics or ratios between harmonics and the fundamental wave (harmonic distortion factor).

## To measure a harmonic of arbitrary order n...

1. Reset the analyzer to ensure that the following operating mode is set: *Sweep Type: Lin. Frequency, Trace - Meas.: S21, Trace - Format: dB Mag*.
2. Open the *Channel - Stimulus* menu and reduce the *Stop* frequency to the maximum frequency of your analyzer divided by n where n is the order of the harmonic you want to measure.
3. Click *Channel - Mode - Harmonics - More Harmonics.* In the dialog opened select the *Harmonic Order* you want to measure.
4. Adjust the *Source* port and the receive port *(Harmonics Measured at)* to your test setup (for a DUT connected between port 1 and 2 of the analyzer you can keep the default settings).
5. *Close* the *More Harmonics* dialog to enable the harmonics measurement and observe the changed default trace *Trc1.*

The trace shows the $n^{th}$ harmonic as a wave quantity $b_2$. The measured quantity is changed automatically when the harmonics measurement is enabled.

*Use 2nd Harmonic Port1 to Port2 and 3rd Harmonic Port1 to Port2 to activate the measurement of the second and third harmonics with a single menu command.*

## To obtain a harmonic distortion factor...

1. Reset the analyzer to ensure that the following operating mode is set: *Sweep Type: Lin. Frequency, Trace - Meas.: S21, Trace - Format: dB Mag*.
2. Open the *Channel - Stimulus* menu and reduce the *Stop* frequency to the maximum frequency of your analyzer divided by n where n is the order of the harmonic distortion factor you want to measure.
3. Click *Channel - Mode - Harmonics - More Harmonics.* In the dialog opened select the *Harmonic Order* you want to measure.
4. Adjust the *Source* port and the receive port *(Harmonics Measured at)* to your test setup (for a DUT connected between port 1 and 2 of the analyzer you can keep the default settings).
5. *Close* the *More Harmonics* dialog to enable the harmonics measurement.
6. Click *Relative Measurement* and observe the new mathematical trace *Trc2.*

The trace belongs to a new channel *Ch2* and shows the $n^{th}$ harmonic distortion factor as a wave quantity $b_2$. With the default trace format the result is displayed in dB units.

# Pulse Profile

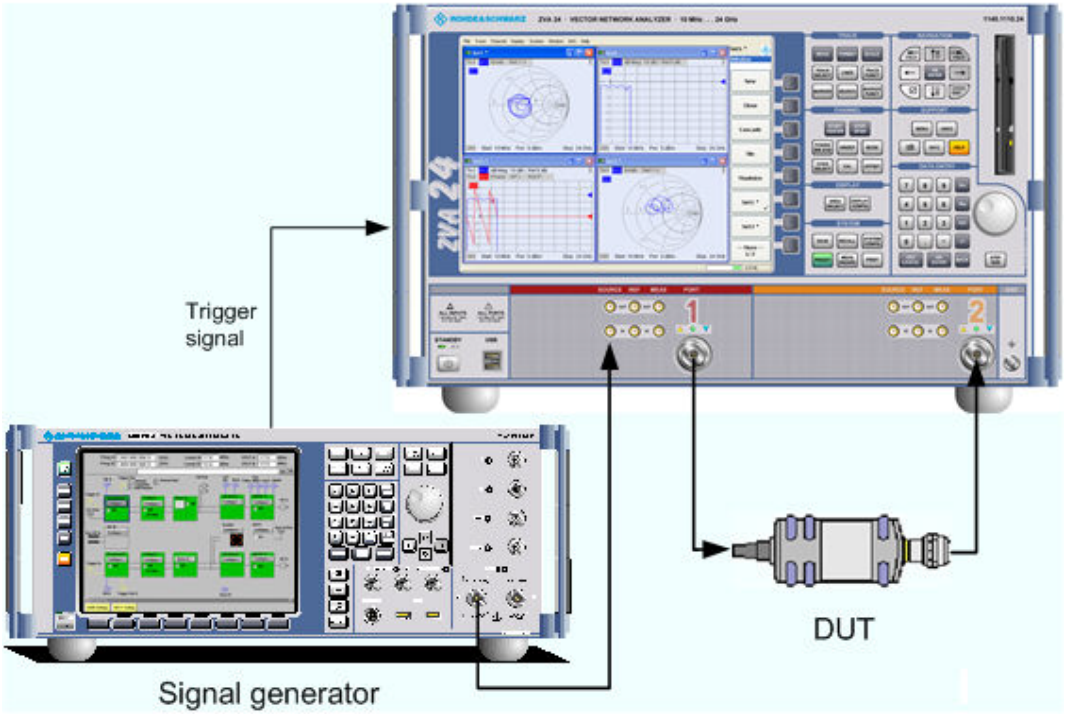
## Pulsed Measurements

With option ZVA-K7, *Pulsed Measurements,* the analyzer provides pulse profile measurements. The pulsed signal must be generated by external means; see introduction to *Pulsed Measurements.*

### Activating a Basic Pulse Profile Measurement

In this example we assume that a pulsed signal (e.g. a signal from an external signal generator) is fed to the SOURCE IN connector of the analyzer port 1 so that it can be used as a port 1 source signal. A DUT is connected between the test ports 1 and 2 of the analyzer; the transmitted pulsed signal is measured at port 2.

We also assume that the signal generator provides a trigger signal which is synchronized to the pulses and applied to the BNC connector EXT TRIGGER on the rear panel of the analyzer. Synchronization of the analyzer and the signal generator by means of a common reference clock is preferable.




The analyzer measures the pulse profile, i.e. the magnitude of the received wave quantity (typically $b_2$) vs. time.

1. Select the appropriate measured quantity and trace format (Trace – Measure – Wave Quantities – b2 Source Port 1 and Trace – Format – dB Mag).
2. Click *Channel – Sweep – Trigger: External* and trigger the measurement on the *Rising Edge* of the trigger signal.
3. Click *Channel – Sweep – Sweep Type – Pulse Profile*.
4. Click *Define Pulse Profile* to access the basic measurement settings.
5. Adjust the *Start* and *Stop* times to the pulse width of your signal: Make sure that *Stop – Start* exceeds the width of a single pulse.

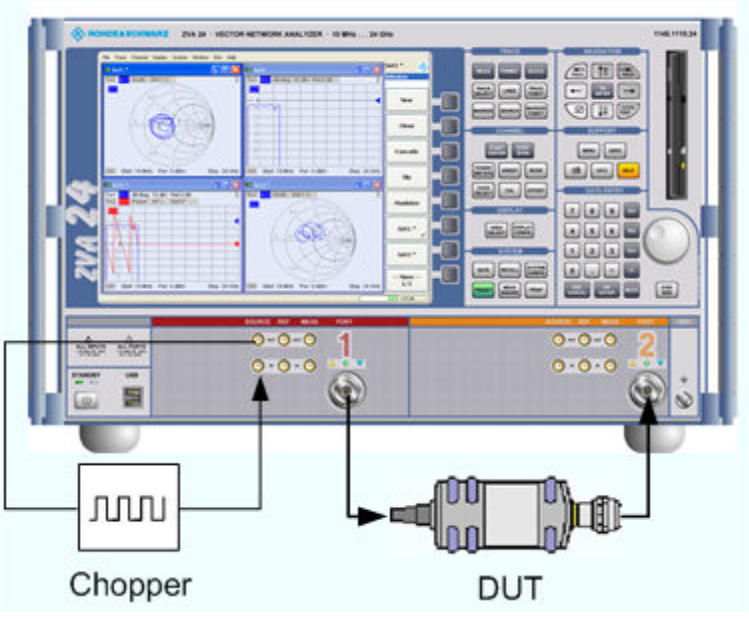
6. Adjust the *Center Freq* to the carrier RF frequency of your pulsed signal.
7. Observe the pulse profile in the diagram area. If desired, vary the *If Bandwidth* and the *Number of Points.*

The x-axis of the active diagram is linearly scaled and labeled with the *Start* and *Stop* times.

Switching off the internal RF generator while an external generator is used can improve the measurement accuracy. *Channel – Power Bandwidth Average – RF Off* also deactivates external generators, so you have to use the settings in the *Source* section of the *Port Configuration* dialog (with option R&S ZVA-K4).

## Measuring a DUT with a Transmission Delay

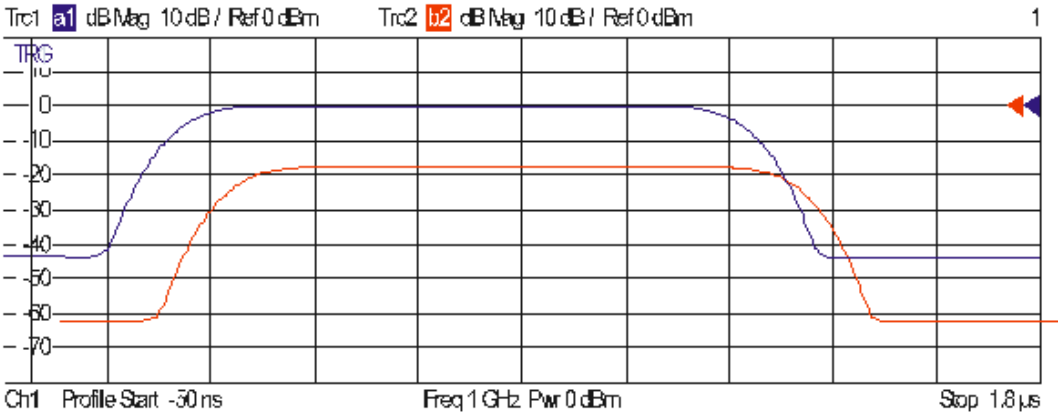
In this example we introduce a chopper between the SOURCE OUT and the SOURCE IN connectors of the analyzer port 1 (see *Front Panel Jumpers)*. The chopper transforms the CW source signal at port 1 into a pulsed signal. In addition a DUT with an attenuation factor of 16 dB and a transmission delay of 100 ns is inserted between port 1 and Port 2 of the analyzer.




The analyzer measures the pulse profile of the reflected and transmitted waves, i.e. the magnitudes of the wave quantities $a_1$ and $b_2$ vs. time and calculates the forward transmission coefficient $S_{21}$.

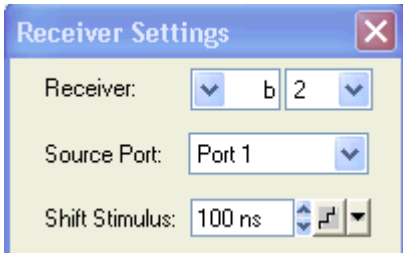
1. Create three traces and select the measured quantities and trace formats *(Trace – Measure – Wave Quantities – a1* etc.*, Source Port 1* and *Trace – Format – dB Mag).*
2. Press *Channel – Sweep – Trigger: External* and trigger the measurement on the *Rising Edge* of the trigger signal.
3. Click *Channel – Sweep – Sweep Type – Pulse Profile*.
4. Click *Define Pulse Profile* to access the basic measurement settings.
5. Adjust the *Start* and *Stop* times to the pulse width of your signal: Make sure that *Stop – Start* exceeds the width of a single pulse.
6. Adjust the *Center Freq* to the frequency of your pulsed signal.
7. Observe the pulse profile in the diagram area. If desired, vary the *If Bandwidth* and the *Number of Points.*

Due to the delay and the attenuation introduced by the DUT, the wave $b_2$ is shifted with respect to $a_1$.as shown below.



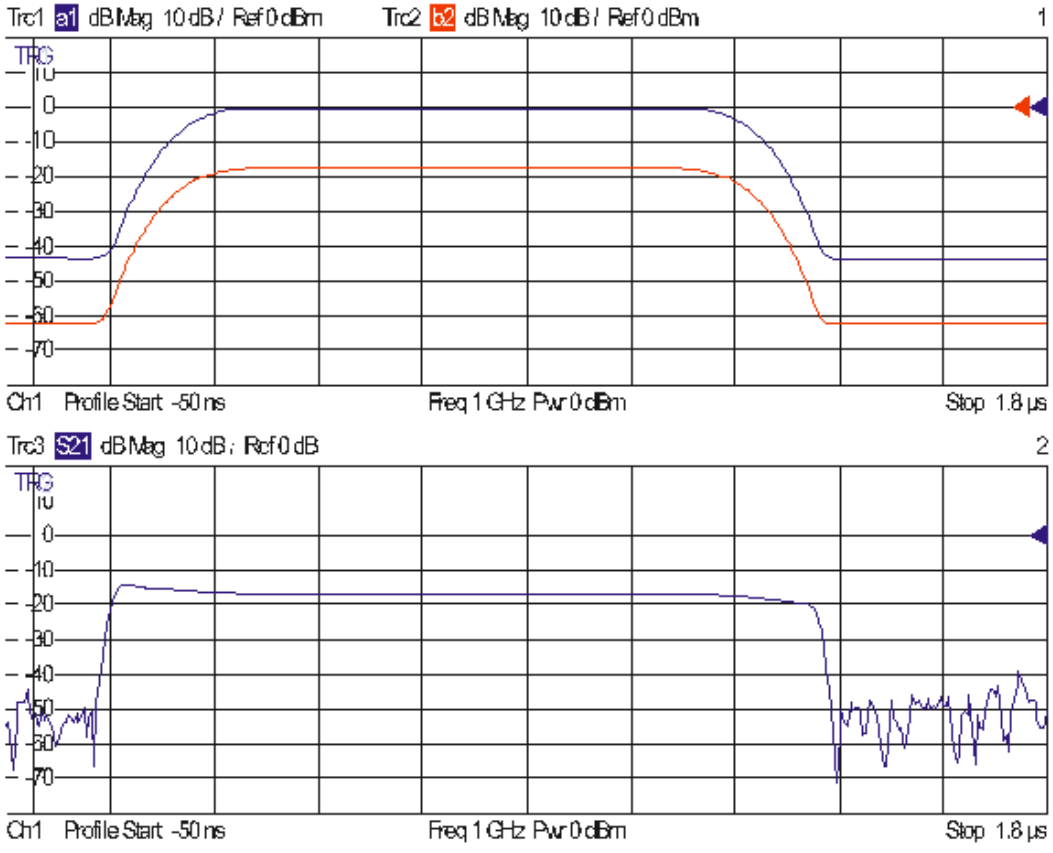

The magnitude of the S-parameter $S_{21}$ is distorted at the beginning and at the end of the pulses because the analyzer divides the two wave quantities at equal times.

8. Re-open the *Define Pulse Profile* dialog and click *Receiver Settings...*
9. In the upper part of the *Receiver Settings* dialog, select the following values:




10. Close the configuration dialogs and observe the offset-corrected traces.

In the diagram area, the wave quantity is shifted so that the $a_1$ and $b_2$ pulses coincide. This also removes the distortions of the S-parameter.


Trc1 a1 dB Mag 10 dB / Ref 0 dBm
Trc2 b2 dB Mag 10 dB / Ref 0 dBm
1
TRG
10
0
-10
-20
-30
-40
-50
-60
-70
Ch1 Profile Start -50 ns
Freq 1 GHz Pwr 0 dBm
Stop 1.8 µs
Trc3 S21 dB Mag 10 dB / Ref 0 dB
2
TRG
10
0
-10
-20
-30
-40
-50
-60
-70
Ch1 Profile Start -50 ns
Freq 1 GHz Pwr 0 dBm
Stop 1.8 µs

# Index


## *

*CLS MERGEFORMAT ..... 5.26

## [

[SENSe<Ch>
]AVERage ..... 6.137
]FREQuency ..... 6.172
]SWEep
POINts ..... 6.221

## 1

12-term error model MERGEFORMAT ..... 3.37

## 2

2-port ..... 4.15
2-port S-parameters ..... 4.15
2-port Z-parameters ..... 4.30

## 7

7-term error model MERGEFORMAT ..... 3.37

## A

a1 ..... 4.15
a'1 ..... 4.18
a2 ..... 4.15
Accessories
Calibration units (ZV-Z5x) ..... 3.41
ACP ..... 4.18
Add Channel ..... 4.221
Add Standard ..... 4.258
Addressed command MERGEFORMAT ..... 5.7
Adjacent Channel Power ..... 4.18
Admittance ..... 3.29
input/output ..... 4.31
transfer ..... 4.31
Admittance ..... 4.31
Alternate ..... 4.218
Area Select ..... 4.270
Attenuators (step) ..... 6.229
Auto (spurious avoidance) ..... 4.219
Automatic calibration ..... 3.41
Autoscale ..... 2.14
Average Factor ..... 6.137
Average On ..... 4.132, 6.137

## B

b1 ..... 4.15
b'1 ..... 4.18
b1/a1 Drive Port 1 ..... 4.18
b2 ..... 4.15
Balanced and Measured Ports ..... 4.177
Balanced port configurations ..... 3.33
Balanced Port Deembedding ..... 4.207
Balanced Port Embedding ..... 4.207
Balance-unbalance conversionMERGEFORMAT ..... 3.32
Balun ..... 3.32
Bandfilter
search ..... 4.110
Bandfilter ..... 4.110
Bandpass Search Ref to Marker ..... 4.110
Bandpass Search Ref to Max ..... 4.110
Bandstop Search Ref to Marker ..... 4.110
Bandstop Search Ref to Min ..... 4.110
Bandwidth ..... 4.132
Basic Concepts MERGEFORMAT ..... 3.1
Block data format MERGEFORMAT ..... 5.11
Boolean parameter MERGEFORMAT ..... 5.11
Brace ..... 6.1
Bracket, square ..... 6.1

## C

Cal group ..... 4.248
Cal pool ..... 4.248
Cal. Ref. ..... 4.221
Calibration ..... 3.36
example ..... 2.4
guided ..... 4.229
measurement examples ..... 9.6
program example MERGEFORMAT ..... 2
wizard ..... 4.229
Calibration kit ..... 3.36
Calibration kit ..... 4.250
Calibration Manager ..... 4.248
Calibration MERGEFORMAT ..... 2
Calibration standard ..... 3.36
Calibration types MERGEFORMAT ..... 3.37
Calibration Unit ..... 3.41
Calibration wizard ..... 2.4
Cartesian diagram ..... 3.19
Cartesian diagrams ..... 4.41
Center ..... 4.129, 6.172
Channel
data flow ..... 3.4
menu ..... 4.129
Sweep ..... 4.138
Channel ..... 4.129
Channel ..... 4.138
Channel Cal ..... 4.248
Channel Manager ..... 4.221
Channel MERGEFORMAT ..... 3.2
Channel settings ..... 3.3, 3.15
Circuit Model ..... 4.258
Close ..... 4.2
Command
common ..... 6.3
common MERGEFORMAT ..... 5.8
device-specific MERGEFORMAT ..... 5.8
processing MERGEFORMAT ..... 5.14
recognition MERGEFORMAT ..... 5.14
sequence MERGEFORMAT ..... 5.14
stucture and syntax MERGEFORMAT ..... 5.8
synchronization MERGEFORMAT ..... 5.14
Command description (notation) ..... 6.1
Common mode MERGEFORMAT ..... 3.34
Complex and real quantities ..... 3.19
Compression point ..... 9.9
CONDition MERGEFORMAT ..... 5.17
Confirmed settings ..... 3.17
Context menu ..... 3.16
Continuous Sweep ..... 4.138

Control menu ... 4.2
Converter (frequency) ... 4.297
Converter control ... 4.297
Couple all ... 4.221
Coupled Markers ... 6.31
Crosstalk ... 4.138
Crosstalk MERGEFORMAT ... 3.37
CW Frequency ... 6.172
CW Time ... 9.3
CW Trigger Events ... 9.3

## D

Data entry MERGEFORMAT ... 2.11
Data Flow
S-parameters ... 3.4
wave quantities ... 3.4
Data Flow ... 3.4
Data processing ... 3.4
Data set MERGEFORMAT ... 5.14
Data trace ... 3.11
Data trace ... 4.58
DC Inputs ... 4.36
DC Inputs MERGEFORMAT ... 8.4
DC MEAS 1 V ... 4.36
DC MEAS 10 V ... 4.36
DC Power ... 4.36
DCL,SDC MERGEFORMAT ... 5.26
Decouple ... 4.221
Deembedding ... 4.207
DEFault MERGEFORMAT ... 5.11
Define Segments ... 4.146, 6.203
Del Error Log ... 4.313
Delete Channel ... 4.221
Delete Segment ... 4.146
Delta marker ... 3.13
Delta Mode ... 6.31
Denominator ... 4.18
Detector ... 4.18
Device message MERGEFORMAT ... 5.7
Device response MERGEFORMAT ... 5.7
Device-specific commands MERGEFORMAT ... 5.8
Diagram
Cartesian ... 3.19
inverted Smith ... 3.23
polar ... 3.20
scaling MERGEFORMAT ... 2.13
Smith ... 3.21
Diagram area ... 4.270
Diagram area MERGEFORMAT ... 3.2, 3.10
Diagram types MERGEFORMAT ... 3.18
Dialogs (general description) MERGEFORMAT ... 3.16
Differential mode MERGEFORMAT ... 3.34
Directivity MERGEFORMAT ... 3.37
Discrete Mode ... 6.31
Dispersion ... 4.263
Display Config ... 4.273
Display formats ... 3.18
Display Line ... 6.8
DOWN MERGEFORMAT ... 5.11

## E

Embedding ... 4.207
ENABle MERGEFORMAT ... 5.17
Error Log ... 4.313
Error queue MERGEFORMAT ... 5.17
Error Terms MERGEFORMAT ... 3.37
ESE MERGEFORMAT ... 5.17
ESR MERGEFORMAT ... 5.17
Evaluation of data ... 2.6
Examples
Reflection measurement MERGEFORMAT ... 2.2
Excursion ... 6.31
Execution
overlapping MERGEFORMAT ... 5.14
sequential MERGEFORMAT ... 5.14

## F

File menu ... 4.4
Fixed Marker ... 6.31
Fixture simulator ... 6.14
Format ... 4.41, 6.13
Free Run ... 4.159
Frequency
sweep ... 4.139
Frequency ... 9.5
Frequency Converter ... 4.297
Frequency Extension ... 4.297
Front Panel Keys
control the GUI MERGEFORMAT ... 2.9
data entry ... 2.11
Full One-Port Calibration MERGEFORMAT ... 3.37

## G

G matrix MERGEFORMAT ... 3.37
Getting started ... 2.1
Global resources ... 3.2
GPIB Address MERGEFORMAT ... 5.3
GPIB bus messages MERGEFORMAT ... 5.7
GPIB Explorer MERGEFORMAT ... 5.3
Guided calibration ... 4.229

## H

Hardcopy ... 6.106
Hardkey bar ... 3.9
Hardware Info ... 4.313
Harmonics ... 4.191
Help menu ... 4.316
Horizontal Line ... 6.8

## I

Ibcntl MERGEFORMAT ... 8.11
Iberr MERGEFORMAT ... 8.11
Ibsta MERGEFORMAT ... 8.11
Iecwin32.exe MERGEFORMAT ... 5.3
IEEE 488.2 MERGEFORMAT ... 5.17
Immediate settings ... 3.17
Impedance
input/output ... 4.30
transfer ... 4.30
Impedance ... 4.30
Impedance MERGEFORMAT ... 3.27
Import/Export Data ... 4.87
INFintiy MERGEFORMAT ... 5.11
Info ... 4.313
Instrument hardware MERGEFORMAT ... 5.14
Interface message MERGEFORMAT ... 5.7
Intermodulation measurement ... 4.18
INV ... 6.1
Inverted Smith diagram ... 3.23
IST flag MERGEFORMAT ... 5.17

## K


Keyboard
on-screen MERGEFORMAT ... 2.11
Keyword (SCPI command) MERGEFORMAT ... 5.8
K-factor ... 4.33


## L


Limit line ... 4.118, 6.16
Lin. Frequency ... 6.221, 9.3
Line (standard) MERGEFORMAT ... 3.37
Linear 2-port
stability ... 4.33
Linear 2-port ... 4.33
LO < RF ... 4.219
LO > RF ... 4.219
Load MERGEFORMAT ... 3.37
Local Max ... 6.31
Local Min ... 6.31
Log. Frequency ... 6.221, 9.3
Lower limit ... 4.118
LXI ... 4.304


## M


Marker ... 3.13
format ... 6.31
menu ... 4.96, 6.31
properties ... 6.31
switch on ... 6.31
Marker ... 4.96
Marker Funct ... 4.116
Marker functions ... 2.14
Marker info field ... 3.14
Match (standard) MERGEFORMAT ... 3.37
Mathematical trace ... 3.11
Mathematical trace ... 4.58
Max Freq ... 4.258
Max Search ... 6.31
Max. Detector ... 4.18
Maximize ... 4.2, 4.314
MAXimum MERGEFORMAT ... 5.11
Meas Delay ... 4.146
MEAS key ... 4.15
Meas. Bandwidth ... 6.138
Meas. Delay ... 6.221
Measure
multiport S-parameters ... 4.15
S-parameters ... 4.15
Measure ... 4.15
Measured quantities ... 3.26
Measurement Speed
Optimizing ... 9.5
Measurement Speed ... 9.5
Measurement Wizard ... 2.1
Measurement Wizard ... 4.309
Memory trace ... 3.11
Memory trace ... 4.58
Menu
structure ... 3.8
Menu bar ... 3.7
Min Freq ... 4.258
Min Search ... 6.31
Minimize ... 4.2
MINimum MERGEFORMAT ... 5.11
Mixed mode parameters MERGEFORMAT ... 3.34
Mixer
scalar measurements ... 4.197
Mixer ... 4.197
Mixer and Harmonics (ZVB-K3) ... 4.191, 4.197
Mixer frequencies ... 6.178
Mode ... 4.164
Mode conversion factorMERGEFORMAT ... 3.34
Modify Standard ... 4.258
More Markers ... 6.31
More S-Parameters ... 4.15
More Y-Parameters ... 4.31
More Z-Parameters ... 4.30
Move ... 4.2
mu1-factor ... 4.33
mu2-factor ... 4.33
Multiport
S-parameters ... 4.15
Multiport ... 4.15
Multiport ... 4.30
Multiport parameters ... 3.27


## N


NAN ... 6.1
NAN (not a number) MERGEFORMAT ... 5.11
Navigation tools (screen) MERGEFORMAT ... 3.6
NCO ... 4.18
Network (standard) MERGEFORMAT ... 3.37
New (setup) ... 4.4
New Channel ... 4.221
Next Channel ... 4.221
NINF (negative infinity) MERGEFORMAT ... 5.11
No Cal ... 4.248
Normal ... 4.15, 4.18, 4.30
Normalization
calibration MERGEFORMAT ... 3.37
NT named pipe MERGEFORMAT ... 5.3
NT pipe A/B MERGEFORMAT ... 5.3
NTR MERGEFORMAT ... 5.17
NTRansition MERGEFORMAT ... 5.17
Number of Points ... 4.138, 4.157, 6.221
Numerator ... 4.18
Numeric suffix ... 6.1


## O


Offset ... 4.263
One Path Two Port Calibration MERGEFORMAT ... 3.37
On-screen keyboard
internal ... 2.12, 3.17
Windows XP ... 2.12
Open (standard) MERGEFORMAT ... 3.37
Open-circuit Z-parameters ... 3.28
OPERation register MERGEFORMAT ... 5.17
Option Info ... 4.313
Options
transfer ... 9.5
ZVAB-K2
Time Domain ... 4.67, 6.65
ZVA-K4
Arbitrary Generator and Receiver Frequencies ... 4.183
ZVA-K6
True Differential Mode ... 4.183
ZVA-K7
Measurements on Pulsed Signals ... 4.150
ZVB-K3
Mixer and Harmonics ... 4.191, 4.197
Output unit MERGEFORMAT ... 5.14
Overlapping command execution MERGEFORMAT ... 5.14

## P


PAE ... 4.36, 6.189
Parameter selection ... 6.186
Peak Search ... 6.31
Points
  Define Segments ... 4.146
  Number ... 4.157
Points ... 4.157
Polar diagram ... 3.20
Port Overview ... 4.207
Power
  Define Segments ... 4.146
Power ... 4.22
Power ... 6.229
Power Added Efficiency ... 4.36
Power Bandwidth Average ... 4.132
Power Calibration ... 4.240
Power sweep ... 4.139, 9.3
PPE MERGEFORMAT ... 5.17
Preset MERGEFORMAT ... 5.26
Print ... 6.106
Printing data ... 2.7
Program example
  calibration MERGEFORMAT ... 2
Program example MERGEFORMAT ... 2
Program Examples MERGEFORMAT ... 1
Proportionality Constant ... 4.36
PTR MERGEFORMAT ... 5.17
Pulse repetition period ... 4.150
Pulse width ... 4.150
Pulsed measurements
  measurement examples ... 9.18
Pulsed measurements ... 4.150


## Q


Query MERGEFORMAT ... 5.7, 5.8
QUEStionable register
  Summary bit MERGEFORMAT ... 5.20
QUEStionable register MERGEFORMAT ... 5.17, 5.20


## R


RatioMERGEFORMAT ... 3.31
Ratios ... 4.18
Ref. Position ... 6.90
Ref. Value ... 6.90
Reference impedance ... 4.207
Reference marker ... 3.13
Reference marker ... 6.31
Reference oscillator ... 6.202
Reference position ... 2.13
Reference value ... 2.13
Reflect (standard) MERGEFORMAT ... 3.37
Reflection masurement
  Example MERGEFORMAT ... 2.2
Remote Control
  special features MERGEFORMAT ... 5.27
Reset Offsets ... 4.263
Reset Values
  Status Reporting System MERGEFORMAT ... 5.26
Reset Values MERGEFORMAT ... 5.3
Resolution bandwidth ... 6.138
Response (to queries) MERGEFORMAT ... 5.8
Restore ... 4.2
Restrict Port Assignment ... 4.258
RMS
  detector ... 4.18
RSDLLibloc MERGEFORMAT ... 8.11
RSDLLibonl MERGEFORMAT ... 8.11
RSDLLibrd MERGEFORMAT ... 8.11
RSDLLibrdf MERGEFORMAT ... 8.11
RSDLLibrsp MERGEFORMAT ... 8.11
RSDLLibsre MERGEFORMAT ... 8.11
RSDLLibtmo MERGEFORMAT ... 8.11
RSDLLibwrt MERGEFORMAT ... 8.11
RSDLLibwrtf MERGEFORMAT ... 8.11
RSDLLilrd MERGEFORMAT ... 8.11
RSDLLilwrt MERGEFORMAT ... 8.11
RSDLLSwapBytes MERGEFORMAT ... 8.11
RSDLLTestSRQ MERGEFORMAT ... 8.11
RSDLLWaitSrq MERGEFORMAT ... 8.11
RSIB address MERGEFORMAT ... 5.3
RSIB interface functions MERGEFORMAT ... 8.11


## S


S11/S12/S21/S22 ... 4.15
Save As ... 4.4
Saving data ... 2.7
SAW Matching Network ... 6.14
Scale Div ... 6.90
Scaling
  Circular diagrams MERGEFORMAT ... 2.14
  Diagrams MERGEFORMAT ... 2.13
Scc/SCD/Sdc/Sdd/Sds/Ssc/Ssd/Sss ... 4.15
SCPI command structure MERGEFORMAT ... 5.8
SCPI commands MERGEFORMAT ... 5.8
SCPI compatibility MERGEFORMAT ... 5.3
SCPI Status Register
  Structure MERGEFORMAT ... 5.17
SCPI Status Register MERGEFORMAT ... 5.17
Screen elements ... 3.6
Search ... 4.103, 6.31
Search Range ... 6.31
Segmented Frequency ... 4.138, 4.146, 9.3
Select Instrument Port ... 4.36
Selectivity
  Define Segments ... 4.146
Selftest ... 4.313
Service Request Enable MERGEFORMAT ... 5.17
Set Frequencies ... 4.197
Setup ... 3.2
Setup Info ... 4.313
Short (standard) MERGEFORMAT ... 3.37
Short-circuit Y-parameters ... 3.30
Single ... 4.163
Single Ended Port Deembedding ... 4.207
Single Ended Port Embedding ... 4.207
Single Sweep ... 4.138
Single-ended mode MERGEFORMAT ... 3.34
Size ... 4.2
Smith diagram ... 3.21
Socket interface MERGEFORMAT ... 5.3
Softkey bar ... 3.8
SOLT MERGEFORMAT ... 3.37
Source Port ... 4.18, 6.221
Source Power ... 4.132, 4.139, 6.229
Span ... 4.129, 6.172
S-Parameter Wizard ... 4.309
S-parameters
  2-port MERGEFORMAT ... 3.26
  data flow ... 3.4
  meaning ... 4.15
  meaning MERGEFORMAT ... 3.26
  measuring ... 4.15
  multiport ... 4.15
  squared amplitudes MERGEFORMAT ... 3.27
S-parameters MERGEFORMAT ... 3.26
S-Params. From ... 4.258
Spurious Avoidance ... 4.219

SRE MERGEFORMAT ..... 5.17
SRQ MERGEFORMAT ..... 5.17
Stability factors ..... 4.33
Start
Define Segments ..... 4.146
Start ..... 4.108, 4.129
Start ..... 6.172
Start Cal ..... 4.225
Start Power Cal ..... 4.240
Status bar ..... 3.9
STatus Byte MERGEFORMAT ..... 5.17
Status Registers MERGEFORMAT ..... 5.17
Status reporting system
Reset Values MERGEFORMAT ..... 5.26
STB MERGEFORMAT ..... 5.17
Step Attenuators ..... 6.229
Step size ..... 3.18
Stimulus
Define Segments ..... 4.146
Stimulus ..... 4.118, 4.129
Stop
Define Segments ..... 4.146
Stop ..... 4.108, 4.129
Stop ..... 6.172
String MERGEFORMAT ..... 5.11
SWEEP key ..... 4.138
Sweep range selection ..... 2.3
Sweep Time ..... 6.221, 9.5
Sweep Type ..... 4.139, 6.186
SWR ..... 4.41
Syntax elements (SCPI, overview) MERGEFORMAT ... 5.11
System Configuration ..... 4.285

## T

Take Cal Sweep ..... 4.197
Target Search ..... 6.31
Target Value ..... 6.31
Text parameter MERGEFORMAT ..... 5.11
Threshold ..... 6.31
Through (standard) MERGEFORMAT ..... 3.37
Time Domain ..... 4.67
Title ..... 3.11
TNA calibration MERGEFORMAT ..... 3.37
TOM calibration MERGEFORMAT ..... 3.37
TOSM calibration MERGEFORMAT ..... 3.37
Trace
data ..... 4.58
data flow ..... 3.4
mathematical ..... 4.58
memory ..... 4.58
Trace Funct ..... 4.58
Trace list ..... 3.12
Trace MERGEFORMAT ..... 3.2
Trace settings ..... 3.3
Trace Statistics ..... 4.80
Traces ..... 3.11
Tracking ..... 4.110, 6.31
Transform ..... 4.67
Transformation Network ..... 4.207
Trigger ..... 4.159, 4.162
Trigger Settings ..... 4.159, 4.162
TRL calibration MERGEFORMAT ..... 3.37
TRM calibration MERGEFORMAT ..... 3.37
True Differential Mode ..... 4.183

## U

Universal command MERGEFORMAT ..... 5.7
UP MERGEFORMAT ..... 5.11
Upper limit ..... 4.118
Upper/lower case ..... 6.1

## V

View menu ..... 4.279
Virtual Differential Mode ..... 4.183
Virtual Transform ..... 4.207, 6.72

## W

Wave Quantity ..... 4.22
Wave quantityMERGEFORMAT ..... 3.30, 3.31
Windows menu ..... 4.314

## Y

Y11/Y12/Y22/Ycc/Ydd/Yss ..... 4.31
Y-parameter ..... 4.31
Y-parameter MERGEFORMAT ..... 3.29

## Z

Z0 ..... 4.30, 4.31
Z11/12/21/22/cc/dd/ss ..... 4.30
Zoom ..... 4.49
Z-parameter ..... 4.25, 4.30
Z-parameter MERGEFORMAT ..... 3.27
ZVAB-K2
Time Domain ..... 4.67, 6.65
ZVA-K6
True Differential Mode ..... 4.183
ZVA-K7
Measurements on Pulsed Signals ..... 4.150
ZVB-K3
Mixer and Harmonics ..... 4.191, 4.197
ZVR compatibility ..... 6.1
ZV-Z51 ..... 3.41
ZV-Z52 ..... 3.41
ZV-Z53 ..... 3.41
ZV-Z58 ..... 3.41