COPPER MOUNTAIN® TECHNOLOGIES

631 E New York Street | Indianapolis, IN 46202 USA
www.coppermountaintech.com

Vector Network Analyzers Using CMT VNA Software

Operating and Programming Manuals

Revision 25.00 03.12.2025

U.S.: +1.317.222.5400
Latin America: +1.9154.706.5920

Singapore: +65.63.23.6546
EMEA: +44 75 03 69 21 13

Contents

Contents

Contents

Contents

Contents

Contents

Contents

Contents

Contents

Introduction

This help system is intended to provide you with all information that is necessary for setup, manual, and remote control of the Copper Mountain Technologies Vector Network Analyzer. Please use navigation tools on the left of the window to access the sections.

If you have any questions or comments, please contact us.

This manual covers the 2-port 1-path VTR models of CMT Analyzers controlled by the CMT VNA software version 1.1.0. The Analyzer models are listed below:

VTR0102

VTR0302

This manual contains design, specifications, functional overview, and detailed operation procedures for the Analyzer, to ensure effective and safe use of its technical capabilities.

Maintenance and operation of the Analyzer should be performed by qualified engineers with basic experience in operating of microwave circuits.

Glossary — The abbreviations which are used in this document.

Web Sites

Copper Mountain Technologies

Safety Instructions

It is highly recommended to follow all safety warnings and precautions provided in this document for operating, servicing, and repairing the Analyzer.

The Analyzer should be used only by skilled and thoroughly trained personnel with the required skills and knowledge of safety precautions.

The Analyzer complies with INSTALLATION CATEGORY II as well as POLLUTION DEGREE 2 as defined in IEC61010–1.

The Analyzer is a MEASUREMENT CATEGORY I (CAT I) device. Do not use the Analyzer as a CAT II, III, or IV device.

The Analyzer has been tested as a stand-alone device and in combination with the accessories supplied by Copper Mountain Technologies, in accordance with the requirements of the standards described in the Declaration of Conformity. If the Analyzer is integrated with another system, compliance with related regulations and safety requirements are to be confirmed by the builder of the system.

Never operate the Analyzer in an environment containing flammable gasses or fumes.

Operators must not remove the cover or any other part of the housing. The Analyzer must not be repaired by the operator. Component replacement or internal adjustment must be performed by qualified maintenance personnel only.

Electrostatic discharge can damage the Analyzer whether connected to or disconnected from the DUT. Static charge can build up on your body and damage sensitive internal components of both the Analyzer and the DUT. To avoid damage from electric discharge, observe the following:

- Always use a desktop anti-static mat under the DUT.
- Always wear a grounding wrist strap connected to the desktop anti-static mat via daisy-chained 1 MΩ resistor.
- Connect the post marked ⏚ on the body of the Analyzer to the common ground of the test station.

All general safety precautions related to operation of electrically energized equipment must be observed.

Definitions of safety symbols used on the instrument and in the manual are listed below.

Refers to the Manual if the instrument is marked with this symbol.

I On (Supply).

O Off (Supply).

 A chassis terminal; a connection to the instrument’s chassis, which includes all exposed metal surfaces.

WARNING

This sign denotes a hazard. It calls attention to a procedure, practice, or condition that, if not correctly performed or adhered to, could result in injury or death to personnel.

CAUTION

This sign denotes a hazard. It calls attention to a procedure, practice, or condition that, if not correctly performed or adhered to, could result in damage to or destruction of part or all of the instrument.

NOTE

This sign denotes important information. It calls attention to a procedure, practice, or condition that is essential for the user to understand.

General Overview

The Vector Network Analyzer is designed for use in the process of development, adjustment, and testing of various electronic devices in industrial and laboratory facilities, including operation as a component of an automated measurement system. The Analyzer is designed for operation with an external PC, which is not supplied with the Analyzer.

The overview of measurement capabilities of the Analyzer is represented in Measurement capabilities.

The block diagram of the Analyzer is represented in Principle of operation.

Specifications

The specifications of each Analyzer model can be found in its corresponding datasheet.

Measurement Capabilities 1-Port

Measured parameters	S11, S21 Absolute power of the incident, reflected or transmitted DUT signals.
Number of measurement channels	Up to 9 channels. Each channel is represented on the screen as an individual channel window. Each channel has its own stimulus signal settings such as frequency range, number of test points, power level, etc.
Data traces	Up to 8 data traces can be displayed in each channel window. A data trace represents one of the following parameters of the DUT: S-parameters, response in time domain, input power response.
Memory traces	Each of the 8 data traces can be saved into memory for further comparison with the current values. Up to 8 memory traces can be created for each data trace.
Data display formats	Logarithmic magnitude, linear magnitude, phase, expanded phase, group delay, SWR, real part, imaginary part, Smith chart format, and polar format.

Sweep setup features

Sweep type	Linear, logarithmic, and segment frequency sweep, when the stimulus power is a fixed value.
Power sweep	Linear power sweep when the frequency is a fixed value.
Measured points per sweep	From 2 to 16,001 for TR1300/1. From 2 to 200,001 for TR5048 and TR7530.

Segment sweep	A frequency sweep within several user-defined segments. Frequency range, number of points, source power, and IF bandwidth can be set for each segment.

Power settings	Source power from -55 dBm to +5 dBm (from -55 dBm to +3 dBm TR1300/1) with resolution of 0.05 dB. In frequency sweep mode the power slope can be set to up to 2 dB/GHz to compensate high frequency attenuation in cables.
Sweep trigger	Trigger modes: continuous, single, hold. Trigger sources: internal, external, bus. The availability of this feature depends on the Analyzer model.

Trace display functions

Trace display	Data trace, memory trace, or simultaneous data and memory traces.
Trace math	Data trace modification by math operations: addition, subtraction, multiplication or division between the data, and memory traces.
Autoscaling	Automatic selection of the scale division and reference level value to have the trace most effectively displayed.
Reference level automatic selection	Automatic selection of the reference level. After selection, the data trace shifts vertically so that the reference level crosses the trace in the middle.
Electrical delay	Linear phase correction according the specified electrical delay.
Phase offset	Phase offset by the specified value in degrees.

Accuracy enhancement

Calibration	Calibration of a test setup (which includes the Analyzer, cables, and adapters) significantly increases the accuracy of measurements. Calibration allows correction of errors caused by imperfections in the measurement system: directivity, source, and load match, tracking, and isolation.
Calibration methods	The following calibration methods of various sophistication and accuracy enhancement are available: • reflection and transmission normalization • full one-port calibration (SOL) • one-path two-port calibration
Reflection and transmission normalization	The simplest calibration method. It provides limited accuracy.
Full one-port calibration (SOL)	Method of calibration performed for one-port reflection measurements. It ensures high accuracy.
One-path two-port calibration	Method of calibration performed for reflection S11 and one-way transmission measurements S21. It ensures high accuracy for reflection measurements, and reasonable accuracy for transmission measurements.
Mechanical calibration kits	It is possible to select one of the predefined calibration kits of various manufacturers or define additional ones.
Electronic calibration modules	Copper Mountain Technologies' automatic calibration modules (ACM's) make Analyzer calibration faster and easier than traditional mechanical calibration and provides the highest accuracy.
Defining of calibration standards	Different methods of calibration standard definition are available: • standard definition by polynomial model • data-based standard (full S-parameter characterization data)

Error correction interpolation	When such settings as start/stop frequencies and number of points are changed, compared to the settings of calibration, interpolation or extrapolation of the calibration coefficients will be applied (Extrapolation is not recommended).
Port Extension	Calibration plane compensation for delay in the test setup.

Supplemental calibration methods

Power calibration	Method of the port power calibration which allows to maintain more stable power levels at the DUT input. The calibration requires connection of an external USB power meter.

Marker functions

Data markers	Up to 16 markers for each trace. A marker indicates the stimulus value and measurement result at a given point of the trace.
Reference marker	Enables indication of any maker value as relative to the reference marker.
Marker search	Search for max, min, peak, or target values on a trace.
Marker search additional features	User-defined search range. Available as either a tracking marker, or as a one-time search.
Setting parameters by markers	Setting of start, stop, and center frequencies from the marker frequency, and setting of reference level by the measurement result of the marker.
Marker math functions	Statistics, bandwidth, flatness, RF filter.
Statistics	Calculation and display of mean, standard deviation and peak-to-peak values of the trace.
Bandwidth	Determines bandwidth between cutoff frequency points for an active marker or absolute maximum. The bandwidth value, center frequency, lower frequency,

	higher frequency, Q value, and insertion loss are displayed.
Flatness	Displays gain, slope, and flatness between two markers on a trace.
RF filter	Displays insertion loss and peak-to-peak ripple of the passband, and the maximum signal magnitude in the stopband. The passband and stopband are defined by two pairs of markers.

Data analysis

Port impedance conversion	This function converts S-parameters measured at the Analyzer's nominal port impedance into values which would be found if measured at arbitrary port impedance.
De-embedding	This function allows mathematical exclusion of the effects of the fixture circuit connected between the calibration plane and the DUT. This circuit should be described by an S-parameter matrix in a Touchstone file.
Embedding	This function allows mathematical simulation of the DUT parameters after virtual insertion of a fixture circuit between the calibration plane and the DUT. This circuit should be described by an S-parameter matrix in a Touchstone file.
S-parameter conversion	This function allows conversion of the measured S-parameters to the following parameters: reflection impedance and admittance, transmission impedance and admittance, and inverse S-parameters.
Time domain transformation	This function performs transformation from frequency domain into response of the DUT to various stimulus types in time domain. Modeled stimulus types: bandpass impulse, lowpass impulse, and lowpass step. Time domain span is set arbitrarily from zero to maximum, which is determined by the frequency steps. Various window shapes allow optimizing the tradeoff between resolution and the level of spurious sidelobes.

Time domain gating	This function mathematically removes unwanted responses in time domain, allowing for measurement of the frequency response without the influence of selected fixture elements. Gating filter types: bandpass or notch. For better tradeoff between gate resolution and the level of spurious sidelobes the following filter shapes are available: maximum, wide, normal, and minimum.

Mixer / converter measurements

Scalar mixer / converter measurements	The scalar method allows measurement of scalar transmission S-parameters of mixers and other devices having different input and output frequencies. No external mixers or other devices are required. The scalar method employs port frequency offset when there is a difference between receiver frequency and source frequency.
Automatic adjustment of frequency offset	This function performs automatic frequency offset adjustment when scalar mixer/converter measurements are performed to compensate for LO frequency inaccuracies internal to the DUT.

Other features

Familiar graphical user interface	Intuitive graphical user interface ensures fast and easy Analyzer operation.
Printout/saving of traces	The traces and data printout function has a preview feature. Previewing, saving, and printing can be performed using MS Word, Image Viewer for Windows, or the Analyzer Print Wizard.
Linux OS support	The Linux version of the analyzer software is designed to run on x86 PCs running Linux. NOTE: Tests must be performed to determine if the Analyzer software is compatible with a particular version of Linux.

Remote control

SCPI	Remote control using textual commands SCPI (Standard Commands for Programmable Instruments). Text messages are delivered over PC networks using HiSLIP or TCP/IP Socket network protocols. VISA Library is recommended to support HiSLIP protocol. The TCP/IP Socket protocol can be supported by the VISA library or directly programmed in any language or environment that supports TCP/IP Sockets. The VISA library is free and widely used software in the field of testing and measurement.

Principles of Operation 1-Port

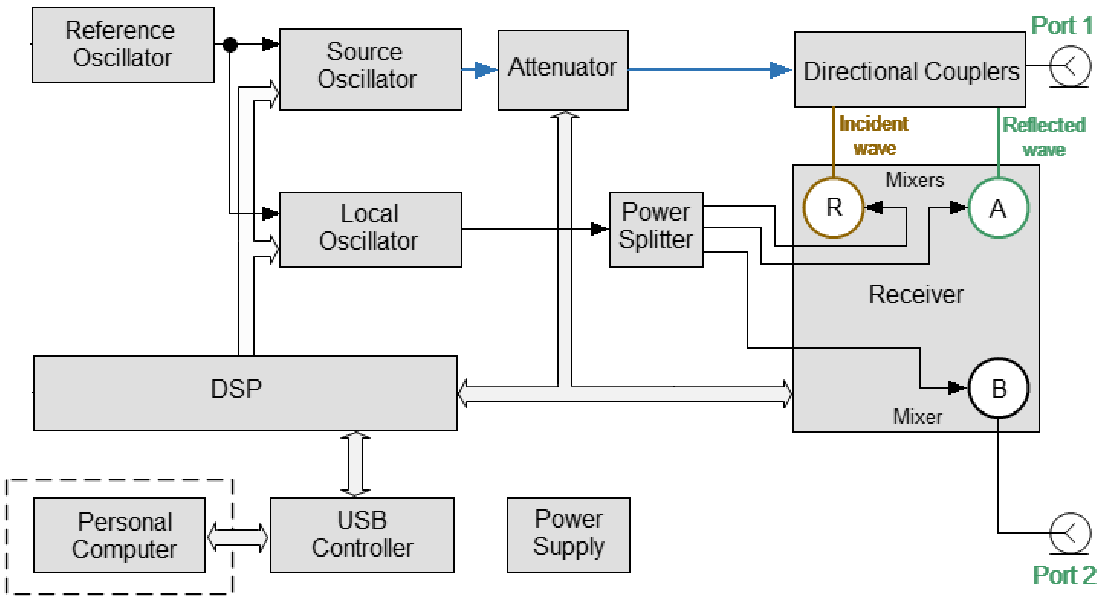
A Vector Network Analyzer (VNA) is a tool for accurate measurement of complex transmission and reflection coefficients (S-parameters) of a Device Under Test (DUT).

The Analyzers described in this manual are USB VNAs. These VNAs consist of an RF measurement module (Analyzer) and supplied processing software — an application which runs on a Windows or Linux based PC or laptop, connected to the Analyzer's hardware via a USB interface. This application controls the RF measurement module, receives and post-processes received raw data and presents the calibrated results to the user in a variety of graphical formats.

For a detailed description of different models of Analyzers see Instrument Series.

The complete specification and supported features list are given in the datasheet of the corresponding Analyzer

The block diagram of the Analyzer is represented in figure below.



The block diagram of the Analyzer

The Analyzer consists of the following functional blocks: a Reference Oscillator, a Source Oscillator, a Local Oscillator, a power control Attenuator, a Power Splitter, Directional Couplers, a three-channel Receiver, a digital signal processor (DSP), a USB Controller and a Power Supply.

A tunable Source Oscillator is the test signal source. The Source Oscillator is based on digital frequency synthesizers. This provides a wide frequency range, set frequency step, and necessary stability for the test signal.

An internal Reference Oscillator provides the Source Oscillator with a stable reference signal.

The Local Oscillator (LO) generates signals using digital frequency synthesizers at an offset from the Source Oscillator which is equal to the Intermediate Frequency (IF) which will be digitized by the VNA IF circuit. The Local Oscillator is the source of the LO signal for the mixers of receiver.

The Power Splitter distributes the LO signal between the three Mixers.

A programmable Attenuator controls the power level of the test signal. This Attenuator is an executive unit of the automatic power control system. For example, when a power calibration has been completed, the Power Correction function uses this Attenuator. Also, the Analyzer can sweep over the output power range at a fixed frequency of test signal using this Attenuator. The user controls the Attenuator by setting the signal power level at the output of the test port. For the power sweep mode, the user sets the range of signal power levels at the output of the measurement port.

After the Attenuator, the test signal passes through the Directional Couplers to Port 1 of the Analyzer. Port 1 is the source of the test signal. The test signal from the signal source goes through the DUT to the connector of the port 2. Port 2 is the receiver of the test signal.

Directional Couplers separate the incident wave and reflected wave of the test signal. The signals from the directional couplers and the signal from the receiver port 2 are supplied into the mixers, where they are converted into first IF 0.3125 MHz (TR5048, TR7530) or 5.037 MHz (TR1300/1), and are transferred further to the 3-Channel receiver:

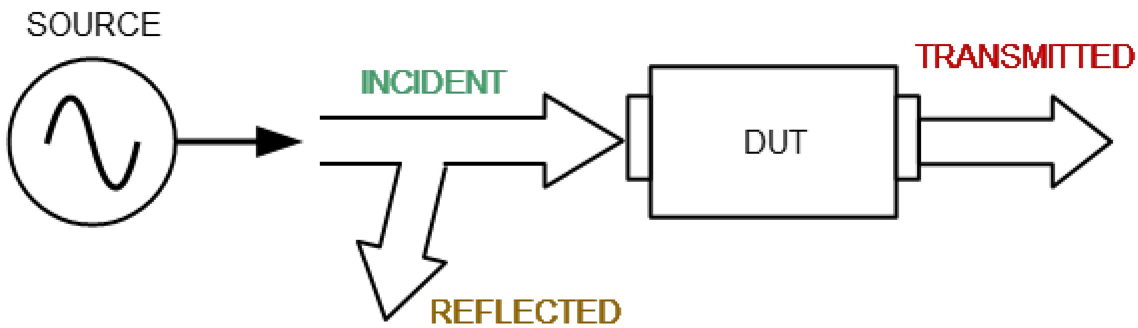
- The reference receiver R processes the incident wave.
- A measuring receiver A processes the reflected wave.
- A measuring receiver B processes the signal transmitted through a DUT to Port 2.

The 3-Channel receiver, after filtration, produces the signal of the second IF, then digitally encodes it and supplies data into the DSP. The DSP performs primary signal processing (filtering, phase difference estimation, magnitude measurement). The user-selected Bandwidth of the second IF is applied by the DSP filter and has a passband from 10 Hz to 30 kHz.

After the primary signal processing, the DSP transmits the information to the control software (TRVNA) running on an external PC. Communication is provided by a USB controller. This software performs the final signal processing and displays the measurement results on the screen of the PC.

Principle of Measuring S-parameters 1-Port

The DUT is connected to the Analyzer ports. The Analyzer emits a test signal (stimulus) out of a source port. Simultaneously, all ports of the Analyzer are receivers. The frequency of the test signal changes in the specified range discretely from point to point. At each frequency point, the Analyzer simultaneously measures the magnitude and phase of the signal transmitted through and reflected from the DUT. These are compared with the magnitude and phase of the incident test signal. The Analyzer calculates the S-parameters of the DUT at each frequency point based on this comparison (See figure below).



S-parameter terms

The S-parameter is a relation between the complex magnitudes of two waves:

$$S_{mn} = \frac{outgoing\ wave\ at\ Port\ m}{incoming\ wave\ at\ Port\ n}$$

Providing the $incoming\ wave\ at\ Port$, except n = 0, where m, n denote the DUT port number.

For a two-port DUT, the full scattering matrix is measured:

$$S = \begin{bmatrix} S11 & S12 \\ S21 & S22 \end{bmatrix}$$

The construction feature of TR Analyzer is that Port 1 is a source port and receiver, and Port 2 is only a receiver (See Principles of Operation) . Thus, the Analyzer can only measure S11 and S21 simultaneously with one DUT connection.

To measure S11 and S21 parameters, connect Port 1 to the input of the DUT, and Port 2 to the output of the DUT. The incident and reflected waves will be measured by Port 1. The transmitted wave will be measured by Port 2.

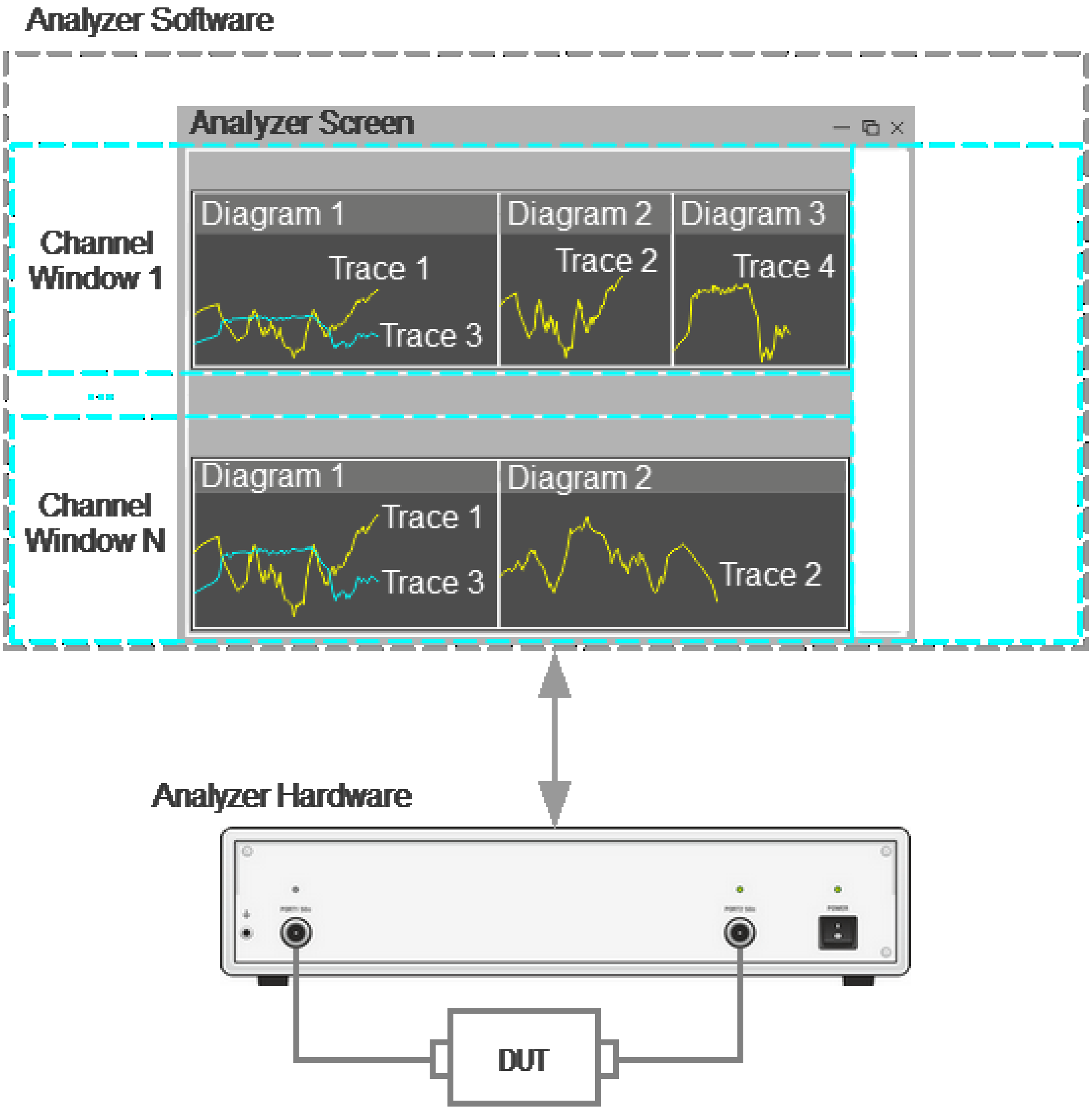
To measure S12 and S22 parameters, reconnect the DUT. Connect Port 1 to the output of the DUT, and Port 2 to the input of the DUT. The incident and reflected waves will be measured by Port 2. The transmitted wave will be measured by Port 1.

This way, using two connections of the DUT, you can measure full scattered matrix of a two-port DUT.

Summarized Description of Hierarchy 1-Port

The following hierarchy of measurement, processing, and display tools is used during operation of the Analyzer (See figure below):

- **Analyzer Hardware** makes radio frequency measurements of the DUT parameters and performs primary processing of measurement results.
- **Analyzer Software** (supplied with the Analyzer) controls the operation of the analyzer components and performs the final mathematical processing and display of the measurement results.



Hierarchy of measuring, processing, and displaying tools

Analyzer Software is displayed as **Analyzer Screen** on the control PC screen, which contains the following:

- **Channel Windows** – the diagram area in which the **Channel** is displayed. For a detailed description of the controls, see Channel Window Layout and Functions.
- **Software and Analyzer Controls**: menu bar, analyzer status bar, and software button bar. For a detailed description of the controls, see Screen Layout and Functions.

Channel – a logical analyzer created by the software to perform DUT measurements with set parameters. The Analyzer control software supports unlimited channels, processing them one at a time. Thus, the same DUT can be sequentially measured by logic analyzers with individual settings.

The channel settings are:

- Sweep Type
- Sweep Range
- Number of Points
- Stimulus Power
- Trigger Settings
- IF Bandwidth Setting
- Calibration and Calibration Kits
- Average Setting

The measurement results of the DUT in the channel are displayed in traces.

Trace – a sequence of data points measured (data trace) or memorized (memory trace) by the analyzer, connected by a line. Each channel contains up to 4 different trace for RVNA and up 16 different traces for RNVNA. The trace is characterized by the following parameters:

- Measurement Parameters
- Format and Scale
- Memory Trace
- Smoothing

The following functions apply to the trace:

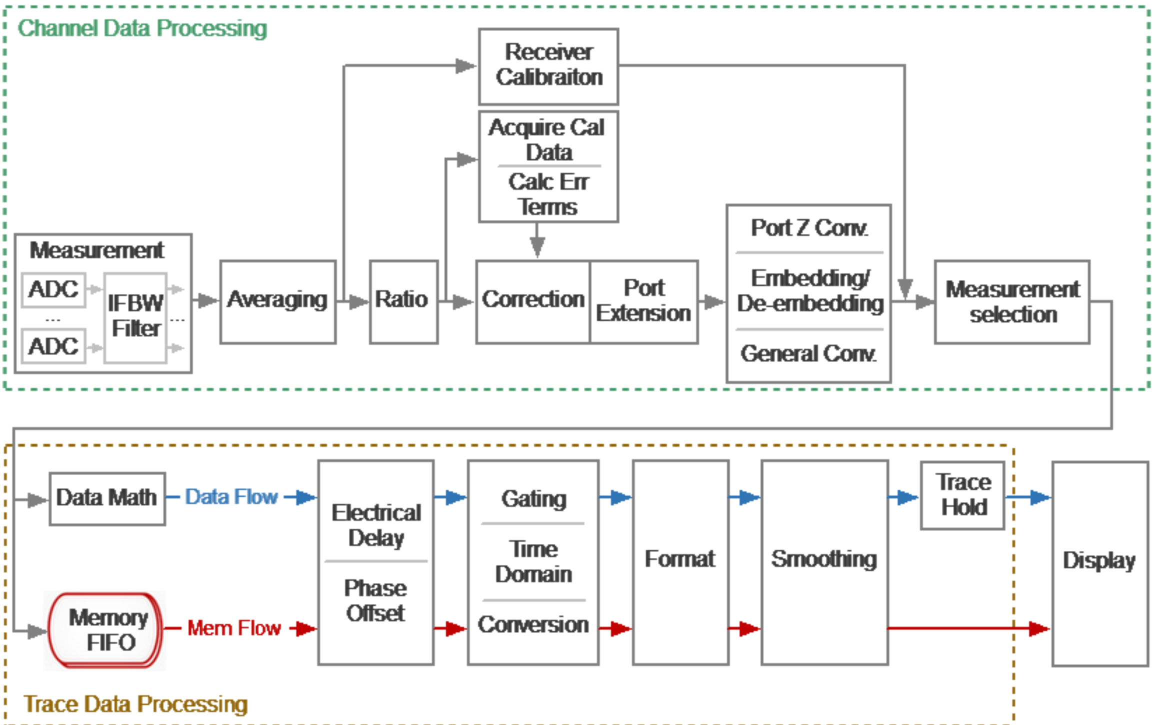
- Markers
- Electrical Delay

- Phase Offset
- Time Domain Gating
-
- Limit Test

Each channel window can display an unlimited number of channels. Convenient placement of traces in the channel window is designated as **Diagram**. Traces can be placed in a single chart or grouped according to user settings in different charts. For a detailed description of working with diagrams, see Trace Allocation.

Internal Data Processing

The following figure shows a flowchart of the Analyzer's internal data processing flow. For a detailed description of remote control access to internal data arrays see in Internal Data Arrays.



Data Processing Flowchart

The Analyzer's internal data processing consists of the following stages:

- **Measurement** is a converting analog signals of receivers into digital ones (receivers R1 and R2 receive the signal of the incident wave, receivers A and B receive the signal reflected from the device under test or passed through the DUT. The received analog measurement signals are converted by ADC (analog-to-digital converters) into digital IF signals and transmitted to the digital processor. The digital processor performs a discrete Fourier transform (DFT) of the IF signals. The analyzer IF bandwidth is equivalent to the bandwidth of the DFT filter. The digital output of each receiver is represented as complex numbers). For more details see Principle of Operation.

- **Averaging** is averaging of the measured data of the receivers for a given number of scan cycles. For a detailed description, see Averaging Setting.
-
- **Ratio** is calculating S-parameters by dividing the complex values of two receiver signals (See The Principle of Measuring S-parameters).
- **Acquire Cal Data** is measuring calibration standards. Complex measured data of all standards are stored in memory. For a detailed description, see Calibration Methods and Procedures.
- **Calc Error Terms** is calculation of calibration coefficients based on measurement data of calibration standards in accordance with the selected calibration method. Calculated calibration coefficients are stored in memory. After calculating the calibration coefficients, the measurement data of the calibration standards is deleted. For a detailed description, see Systematic Errors.
- **Correction** is an application of calibration coefficients to raw S-parameters. At this stage, systematic measurement errors introduced by the analyzer and the measuring setup are eliminated. For a detailed description, see Calibration Methods and Procedures.
- **Port extension** is a fixture simulation in which the addition or removal of a transmission line of a given length for each test port is mathematically simulated. This allows to offset the calibration reference plane by the length of the line. For a detailed description, see Port Extension.
- **Port Z Conv** is the fixture simulation to convert S-parameter when the reference impedance is changed to an arbitrary impedance value. See Port Reference Impedance (Z) Conversion.
- **De-embedding** is the fixture simulation to eliminate the influence of a certain circuit from the measurement results. See De-embedding.
- **Embedding** is the fixture simulation for embedding some virtual circuit in the measured circuit. See Embedding.
- **Measurement Selector** is a selection of display of measured S-parameter or absolute (receiver) data. Data for the trace is selected from a matrix of corrected S-parameters or corrected receiver data. See Measurement Parameters Settings.
- **Memory FIFO** is copying current measurements to memory (S-parameter or receiver data). The software contains a set of cells for storing measurements (memory). It is possible to record up to 8 of these saves. In this case, if all 8 saves are occupied, then the next save will delete the save created by the very

first in time. Further, the memory data is processed in parallel with the measured data. See Memory Trace Function.

- **Data Math** — mathematical operations between measured data and data in memory. When using FIFO memory, the operation is performed with active memory. Available functions: add measured data to memory data, subtract memory data from measured data, multiply/divide measured data by memory data. The result of the operation replaces the measured data. See Memory Trace Function.
- **Electrical Delay** is the compensation of the electrical delay of the DUT when measuring the trace. Unlike port extension, the method is applied individually for each trace. See Electrical Delay Setting.
- **Phase Offset** is setting a constant phase offset of the trace. See Phase Offset Setting.
- **Time Domain** is conversion of the measured S-parameter in the frequency domain into the response of the circuit under investigation in the time domain. See Time Domain Transformation.
- **Gating** is a removal of unwanted responses in the time domain. See Time Domain Gating.
-
- **Format** is selection of the display format of the measured data on the trace. See Format Setting.
- **Smoothing** is an averaging of adjacent points of the trace by a moving window. See Smoothing Setting.
- **Trace Hold** is holding the maximum or minimum values of the trace. See Trace Hold.
- **Display** — data processing for displaying on the screen in the form of a trace of a given format. Scaling is applied to the traces according to the data format, according to selected reference line position and value and scale/grid settings. See Channel Window Layout and Functions.

Instrument Series

This section describes the different series and models of Analyzers.

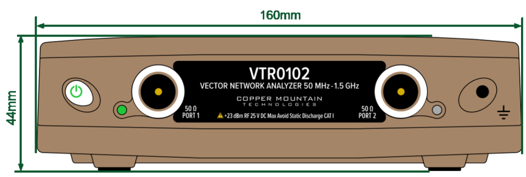
The front and rear panels of each Analyzer are shown further in this section, along with the controls located on those panels.

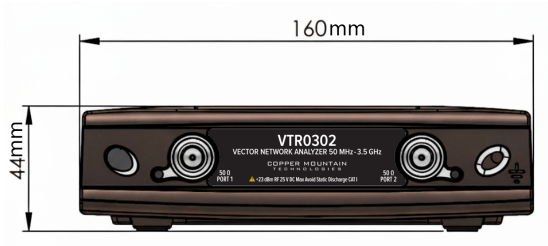
Models of TR VNA:

- VTR0102
- VTR0302

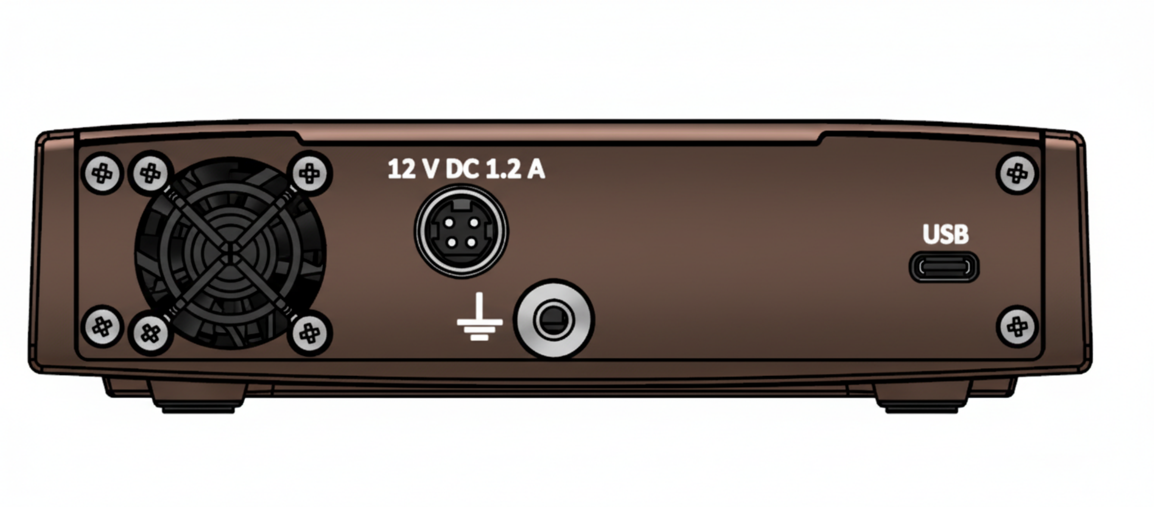
VTR0102 and VTR0302

The front and rear panel view of the Analyzers is represented in the figures below.





VTR0102 and VTR0302 Front Panel



VTR0102 and VTR0302 Rear Panel

Parts of the VTR0102 and VTR0302

Power Button

Switches the Analyzer ON and OFF.

The Analyzer can be turned ON/OFF at any time. The VNA loads its operating firmware from the PC each time upon powering up. The process will take approximately 10 seconds, after which the Analyzer will be ready for operation.

Test ports

The type-N 50 Ω test port 1 and test port 2 are intended for DUT connection. Test port 1 is used as a source of the stimulus signal and receiver of the incident and reflected wave signals. Test port 2 is used as a receiver of the response signal of the DUT.

If the DUT is connected to test port 1 of the Analyzer, it is possible to measure the reflection parameter S11 of the DUT. If the DUT is connected to both test ports of the Analyzer, it is possible to measure S11 and S21 of the DUT.

Each test port has an LED indicator. A test port can be used either as a source of the stimulus signal or as a receiver of the response signal from the DUT. The stimulus signal can only appear on the only one port at a time.

USB Connector

The type C USB 2.0 port is intended for connection to USB port of the personal computer via the supplied USB cable.

Ground Terminal

To avoid electric shock, use this terminal for grounding.

The Ground terminal allows to connect the body of the Analyzer directly to the test station ground in order to ensure electrical safety. Located on both the front and the back panels.

Power Cable Receptacle

The power supply receptacle is intended for an external DC power supply voltage of 12V.

Preparation for Use

Unpack the Analyzer and other accessories.

CAUTION	Please keep packaging to safely ship the instrument for annual calibration!

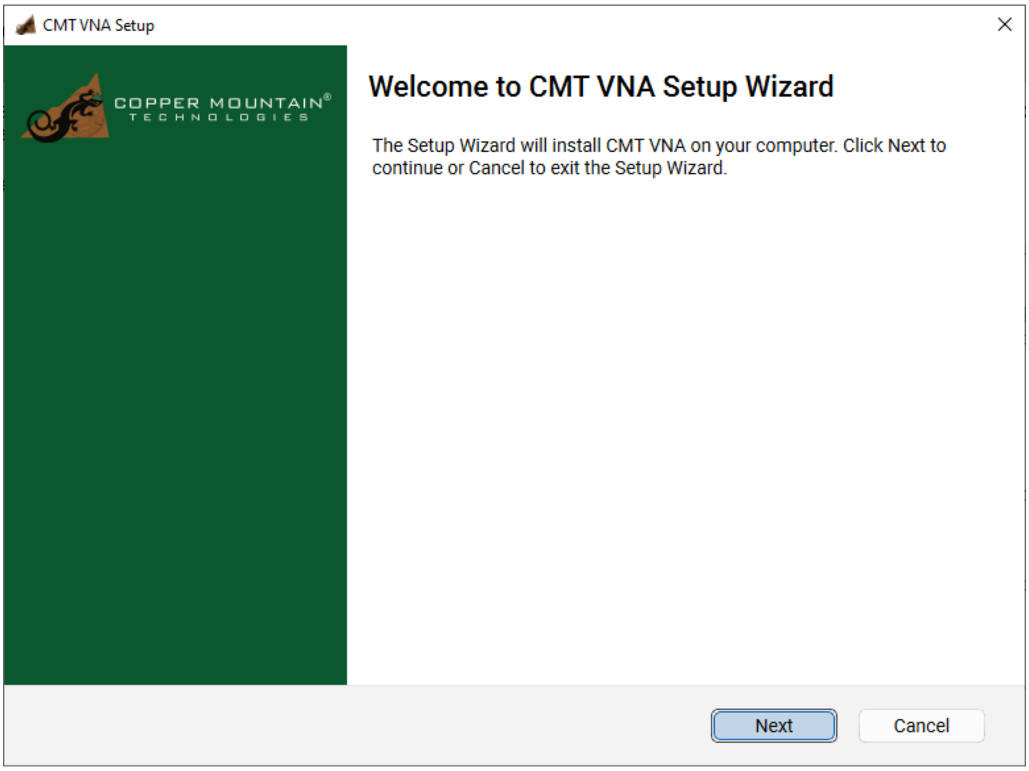
Software Installation

Minimal system requirements	x86 compatible PC running WINDOWS or LINUX WINDOWS 107 or higher LINUX Ubuntu 24.04 or higher 8 GB RAM Minimum USB 2.0 High Speed

Windows installation procedure

Find the Analyzer software installer file cmtvna-setup-X.X.X.exe in the shipped flash-drive or download it from www.coppermountaintech.com. Where X.X.X stands for version number.

Run the cmtvna-setup-X.X.X.exe installer file. Follow the instructions of the installation wizard.



Installation wizard

Linux installation procedure (Example for Raspberry Pi OS based on Debian 13 Trixie)

1. Download the analyzer software file cmtvna-linux-aarch64.tar.gz for ARM based processer (cmtvna-linux-x86_64.tar.gz if processor is based on x86) from https://coppermountaintech.com.
2. Download the libicu74.2 source code from https://github.com/unicode-org/icu/archive/refs/tags/release-74-2.tar.gz and extract this file.
3. In the extracted content, go to /icu4c/source and run these lines to compile the libicu74:

```
./runConfigureICU Linux --prefix=/opt/icu74
make -j$(nproc)
sudo make install
```

Operating and Programming manual is the same as for the Windows application. Download it from www.coppermountaintech.com.

4. Create the USB rules for CMT VNAs by running:

```
sudo ./install_vna_rules.sh
```

When running CMTVNA, run these lines:

```
export LD_LIBRARY_PATH=/opt/icu74/lib:$LD_LIBRARY_PATH
./cmtvna
```

Getting Started

This section is organized as a sample session of the Analyzer. It describes the main techniques for measurement, for example, measuring the reflection coefficient parameters of the DUT. SWR and reflection coefficient phase of the DUT will be analyzed.

In this example, only one test port of the Analyzer is used for reflection coefficient measurement. The instrument sends the stimulus to the input of the DUT and then receives the reflected wave. If the DUT is a two-port device, its unused port should be terminated with a LOAD standard. The results of these measurements can be represented in various formats.

A typical setup for reflection coefficient measurement is shown below.

Reflection Measurement Circuit

To measure SWR and reflection coefficient phase of the DUT in the given example, go through the following steps:

- Prepare the Analyzer for reflection measurement.
- Set stimulus parameters (frequency range, number of points).
- Set IF bandwidth.
- Set the number of traces to 2, assign measured parameters and display format to the traces.
- Set the scale of the traces.
- Perform a one-port calibration.
- Analyze SWR and reflection coefficient phase using markers.

NOTE	The Analyzer can be controlled via softkey panel located on the right-hand part of the screen. The analyzer also allows to perform quick control by the mouse (See Quick Setting Using a Mouse).

Analyzer Preparation for Reflection Measurement

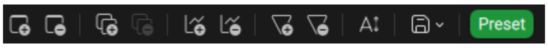
Turn on the Analyzer and warm it up for the period of time stated in its specifications (40 minutes typically).

READY STATE FEATURES	The bottom line of the screen displays the instrument status bar. It should read **Ready**.

Connect the DUT to the test port of the Analyzer. Use the appropriate adapters for connection of the DUT input to the Analyzer test port if it is impossible to connect the DUT input to the test port directly..

Analyzer Presetting

Before starting the measurement session, reset the Analyzer to the initial condition. The initial condition setting is described in Default Settings Table.

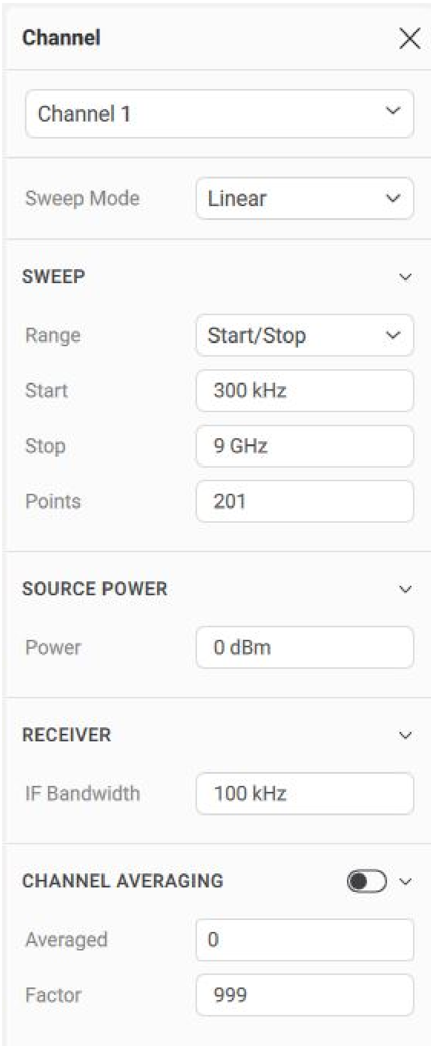


To restore the initial condition of the Analyzer, press the Preset button in the top toolbar.

Channel Settings

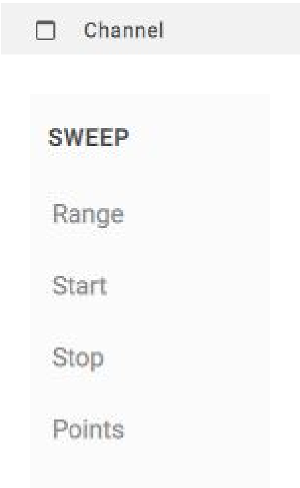
The Channel submenu contains the following parameters:

- Sweep Mode
- Sweep (Range, Start, Stop, Points)
- Source Power
- Receiver
- Channel Averaging



Settable parameters in the channel menu

For the current example, set the frequency range from 10 MHz to 3 GHz.



To set the start frequency of the frequency range to 10 MHz, navigate to:

Channel > Sweep > Start

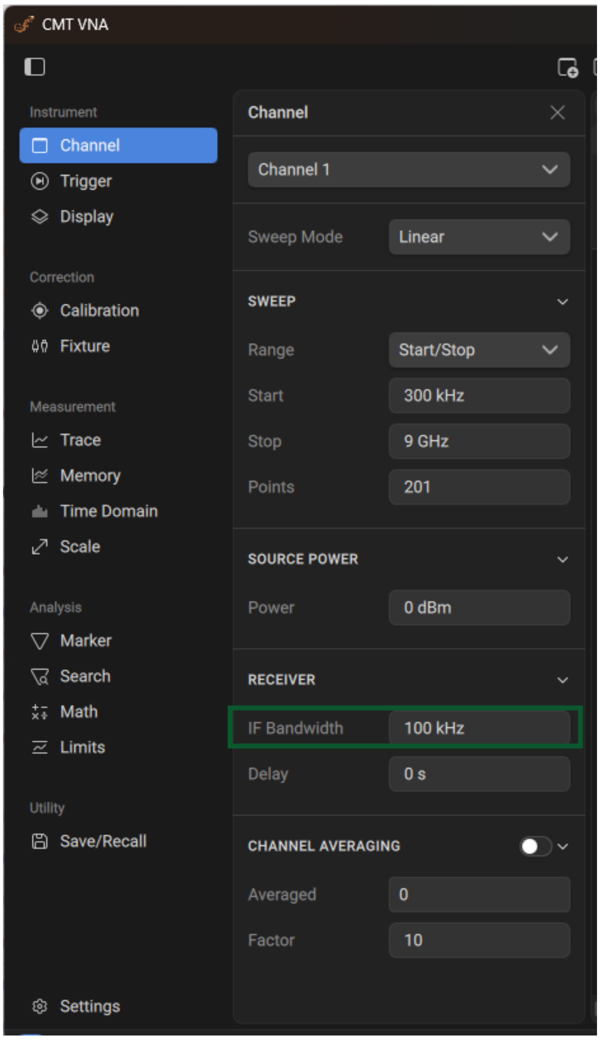
To set the stop frequency of the frequency range to 3 GHz, use the following softkeys:

Channel > Sweep > Stop

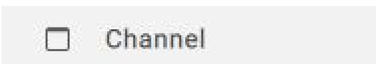
NOTE The **Start** and **Stop** values of the frequency range can be set using the mouse (See Sweep Start Setting).

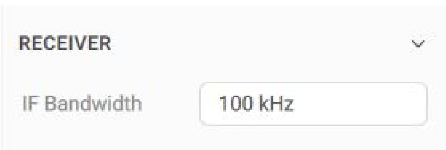
IF Bandwidth Setting

For the current example, set the IF bandwidth to 3 kHz.



IF bandwidth value in the channel status bar





To set the IF bandwidth, navigate to:

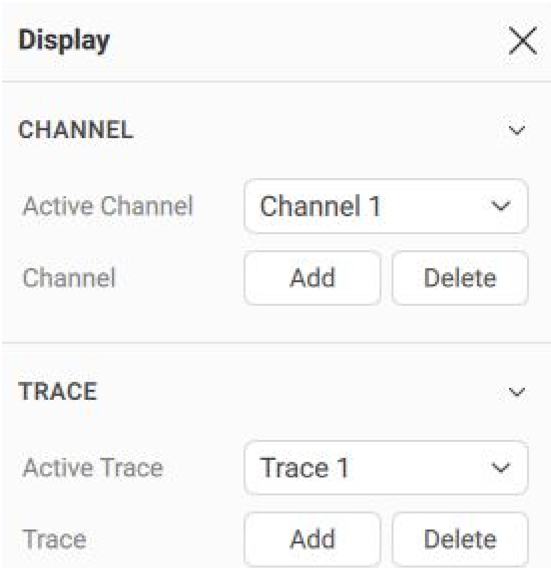
Channel > Receiver > IF Bandwidth

Then enter the desired bandwidth in the textbox and press enter.

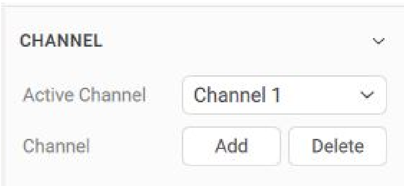
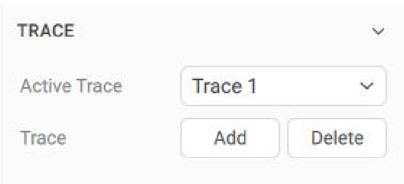
NOTE IF bandwidth can be set using the mouse (See IF Bandwidth Setting).

Number of Traces, Measured Parameter and Display Format Setting

The Display menu can be used to add or remove channels and traces, as well as select the active channel and active trace.



Display Submenu

	To set the active channel, use the drop down menu: **Display > Channel > Active Channel**
	To add or delete a channel, use the Add or Delete buttons: **Display > Channel > Add/Delete**
	To set the active trace, use the drop down menu: **Display > Trace > Active Trace** Activate the trace before assigning the measurement parameters.
	To add or delete a trace, use the Add or Delete buttons:

Display > Trace > Add/Delete

This can also be done in the Trace menu.

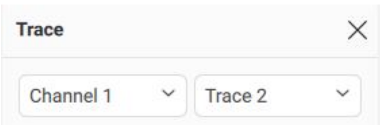
Activate the trace before assigning the measurement parameters.

To assign a parameter to the trace:

Trace > Measurement > Trace

To set the active trace display format, use the Format drop down:

Format > SWR



To set the display format for another trace, use the drop down menu at the top of the Trace submenu.

Trace Scale Setting

For convenience of operation, change the trace scale using automatic scaling function (See Automatic Scaling).

To set the scale of the active trace using the auto-scaling function, use the Autoscale button in the top toolbar.

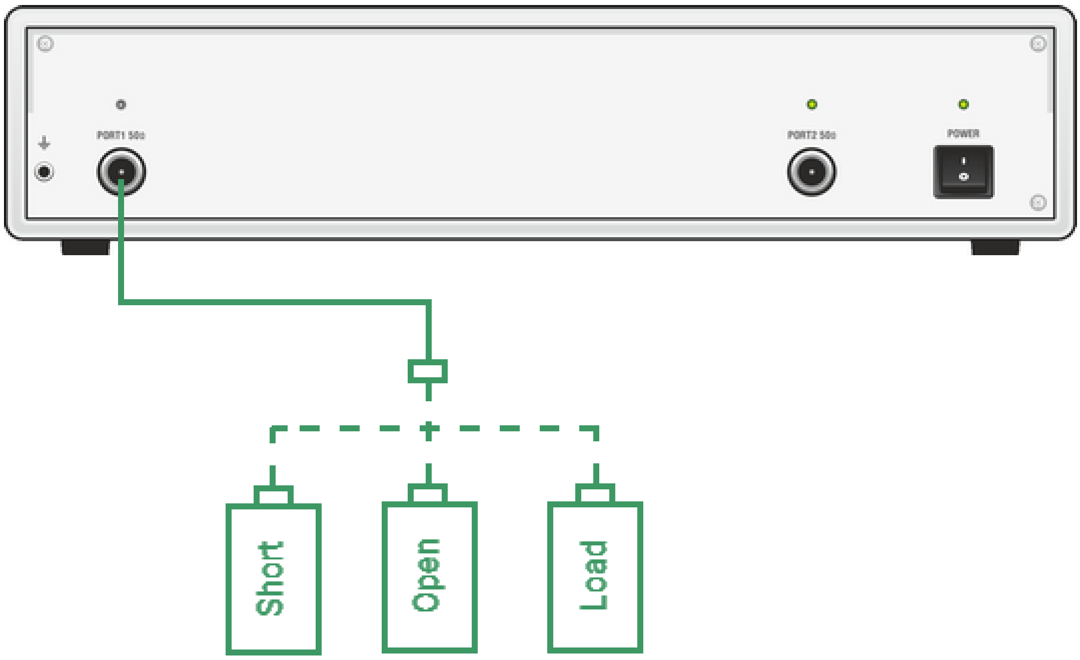
NOTE	The trace scale can also be set manually using the softkeys or using the mouse (Setting with softkeys is described in Rectangular Scale, setting by mouse in Trace Scale Setting).

Analyzer Calibration for Reflection Coefficient Measurement

Calibration of the entire measurement setup — which includes the Analyzer, cables, and adapters involved for the DUT connection — greatly enhances the accuracy of the measurement.

To perform full one-port calibration, prepare the kit of calibration standards: OPEN, SHORT, and LOAD. To perform proper calibration, select the correct kit type in the software. This kit contains a description and specifications of the standards

To perform full one-port calibration, connect calibration standards to the test port one after another and perform the measurement, as shown below.



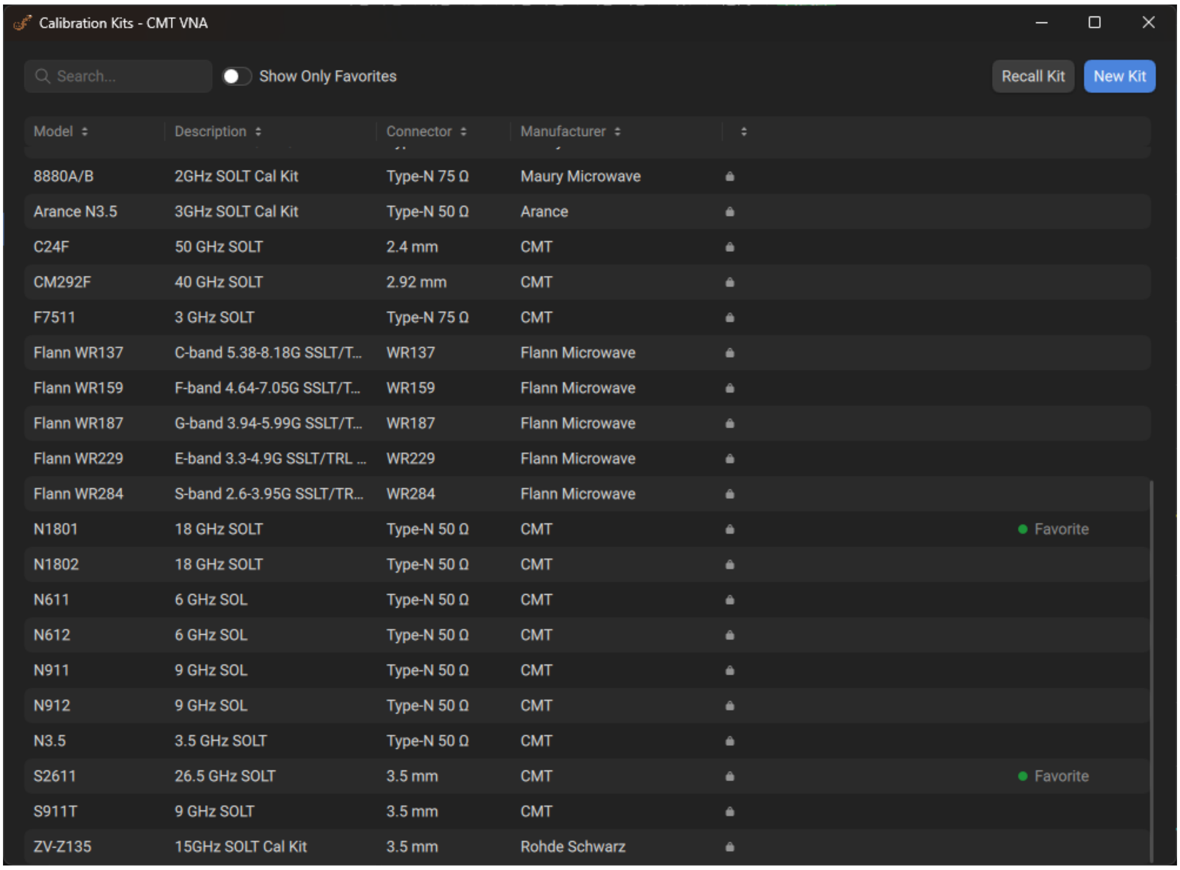
Full one-port calibration circuit

To select the calibration kit, use the following softkeys:

Calibration > Edit Kits

Use the "..." on the right-hand side of the window that opens to enable/favorite the desired cal kit(s), or use the search bar in the top left to search for the desired kit and then use the "..." to enable the kit.

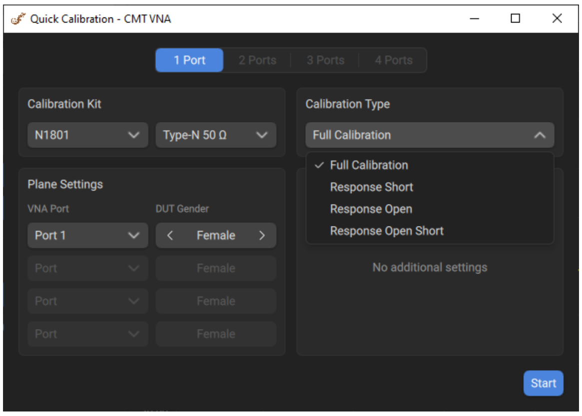
You must enable a kit in order for it to show up in the Quick Calibration Wizard.



Model	Description	Connector	Manufacturer	
8880A/B	2GHz SOLT Cal Kit	Type-N 75 Ω	Maury Microwave	
Arance N3.5	3GHz SOLT Cal Kit	Type-N 50 Ω	Arance	
C24F	50 GHz SOLT	2.4 mm	CMT	
CM292F	40 GHz SOLT	2.92 mm	CMT	
F7511	3 GHz SOLT	Type-N 75 Ω	CMT	
Flann WR137	C-band 5.38-8.18G SSLT/T...	WR137	Flann Microwave	
Flann WR159	F-band 4.64-7.05G SSLT/T...	WR159	Flann Microwave	
Flann WR187	G-band 3.94-5.99G SSLT/T...	WR187	Flann Microwave	
Flann WR229	E-band 3.3-4.9G SSLT/TRL ...	WR229	Flann Microwave	
Flann WR284	S-band 2.6-3.95G SSLT/TR...	WR284	Flann Microwave	
N1801	18 GHz SOLT	Type-N 50 Ω	CMT	Favorite
N1802	18 GHz SOLT	Type-N 50 Ω	CMT	
N611	6 GHz SOL	Type-N 50 Ω	CMT	
N612	6 GHz SOL	Type-N 50 Ω	CMT	
N911	9 GHz SOL	Type-N 50 Ω	CMT	
N912	9 GHz SOL	Type-N 50 Ω	CMT	
N3.5	3.5 GHz SOLT	Type-N 50 Ω	CMT	
S2611	26.5 GHz SOLT	3.5 mm	CMT	Favorite
S911T	9 GHz SOLT	3.5 mm	CMT	
ZV-Z135	15GHz SOLT Cal Kit	3.5 mm	Rohde Schwarz	

Calibration kits list

To perform full one-port calibration (SOL), execute measurements of the three standards in turn. After completion, the table of calibration coefficients will be calculated and saved into the memory of the Analyzer. Before starting calibration, disconnect the DUT from the Analyzer.



To perform full one-port calibration, use the following softkeys:

Calibration > Open Wizard

Select the desired calibration kit and set the gender of the DUT on Port 1. Then click **Start**.

Connect a SHORT standard and click **Measure.**

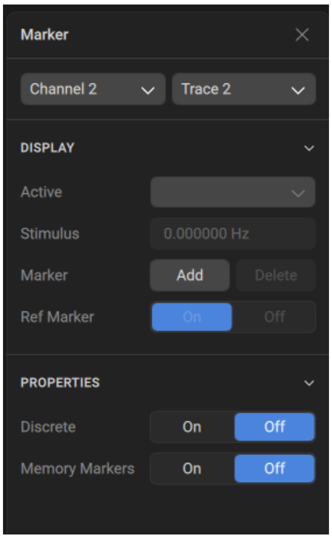
Connect an OPEN standard and click **Measure**.

	Connect a LOAD standard and click **Load**. To complete the calibration procedure and calculate the table of calibration coefficients, click the **Apply** softkey.

Connect the DUT to the calibrated Analyzer port again after calibration is done.

SWR and Reflection Coefficient Phase Analysis Using Markers

This section describes how to determine the measurement values at three frequency points using markers. The Analyzer screen view is shown the screen shot below. In the current example, a reflection standard of SWR = 1.2 is used as a DUT.



To create a new marker, use the Add Marker button in the toolbar  or use:

Marker > Add

A new marker is placed in the center of the stimulus axis and assigned to be active. To edit just created marker:

Marker > Display > Stimulus

Then enter the frequency value in the input field in the graph, e.g. to enter frequency 200 MHz, press «2», «0», «0» and «M» keys on the keypad.

Repeat the above procedure three times to enable three markers at different frequency points.

NOTE	For more details on working with markers, see Markers, Markers, Marker Stimulus Value Setting.

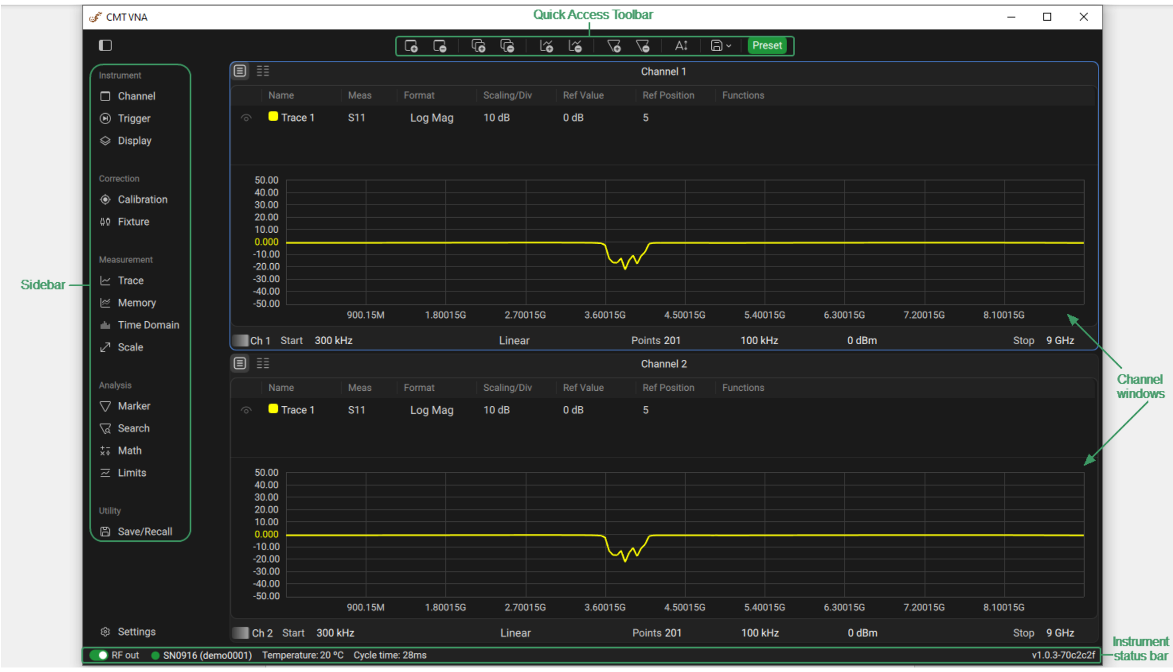
User Interface

The software is displayed as an Analyzer window on the PC screen. The Analyzer window contains:

- Sidebar and Quick Access Toolbar to control the Analyzer.
- Channel windows to display measurement results in the form of traces and numerical values.
- Instrument status bar to display information about the state of the Analyzer.

A detailed description of the software window elements is given further in this section.

An example of the Analyzer window, with the main elements highlighted, is shown in the figure below.

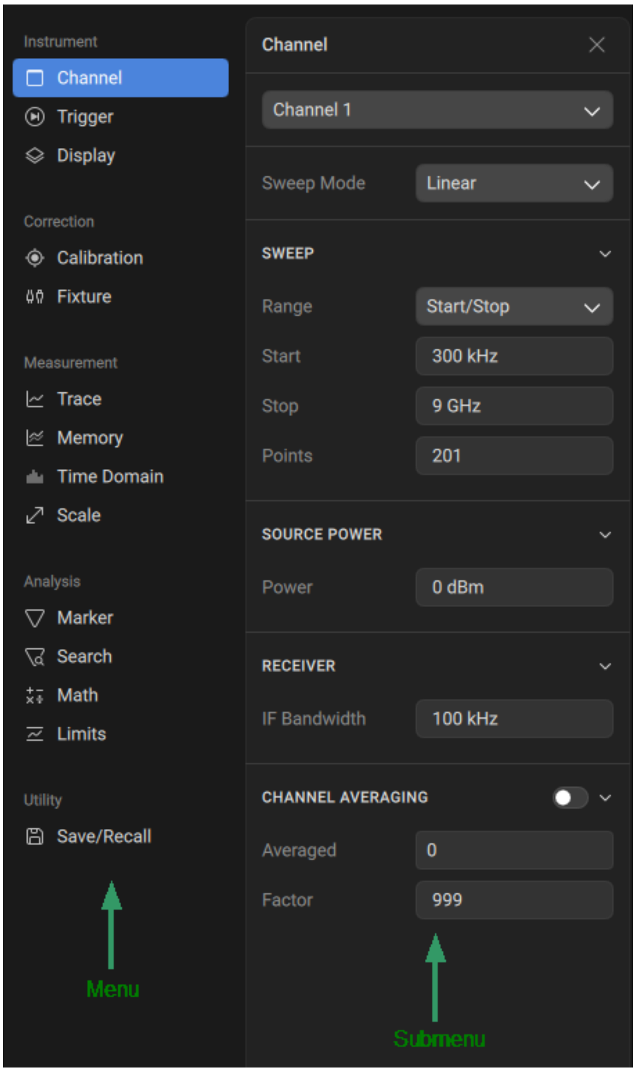


Analyzer Window

Sidebar

The sidebar is located along the left side of the Analyzer window and allows easy access to all software functions.

The sidebar consists of 2 levels: the main menu and the submenu (see figure below).



Sidebar

The main menu contains buttons corresponding to the functions of the Analyzer. Click on the function button in the main menu to expand or collapse its submenu (see figure below). By default, the submenu is hidden.

NOTE

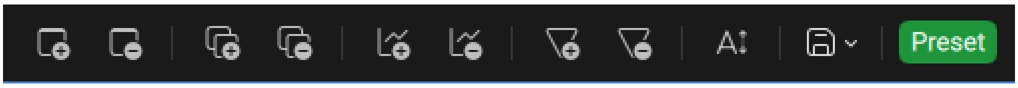
If necessary, the submenu can be closed by clicking the  icon in the right upper corner of the submenu.

If necessary, the main menu can be expanded or collapsed. To do this, click on the icon located at the top of the main menu (see figure below).

NOTE	Collapse the main menu to increase the size of the channel windows.

Quick Access Toolbar

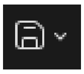
The quick access toolbar is located along the top of the Analyzer window and contains the buttons for the most frequently used functions (see figure below).



Quick Access Toolbar

Description of the buttons is given in the table below.

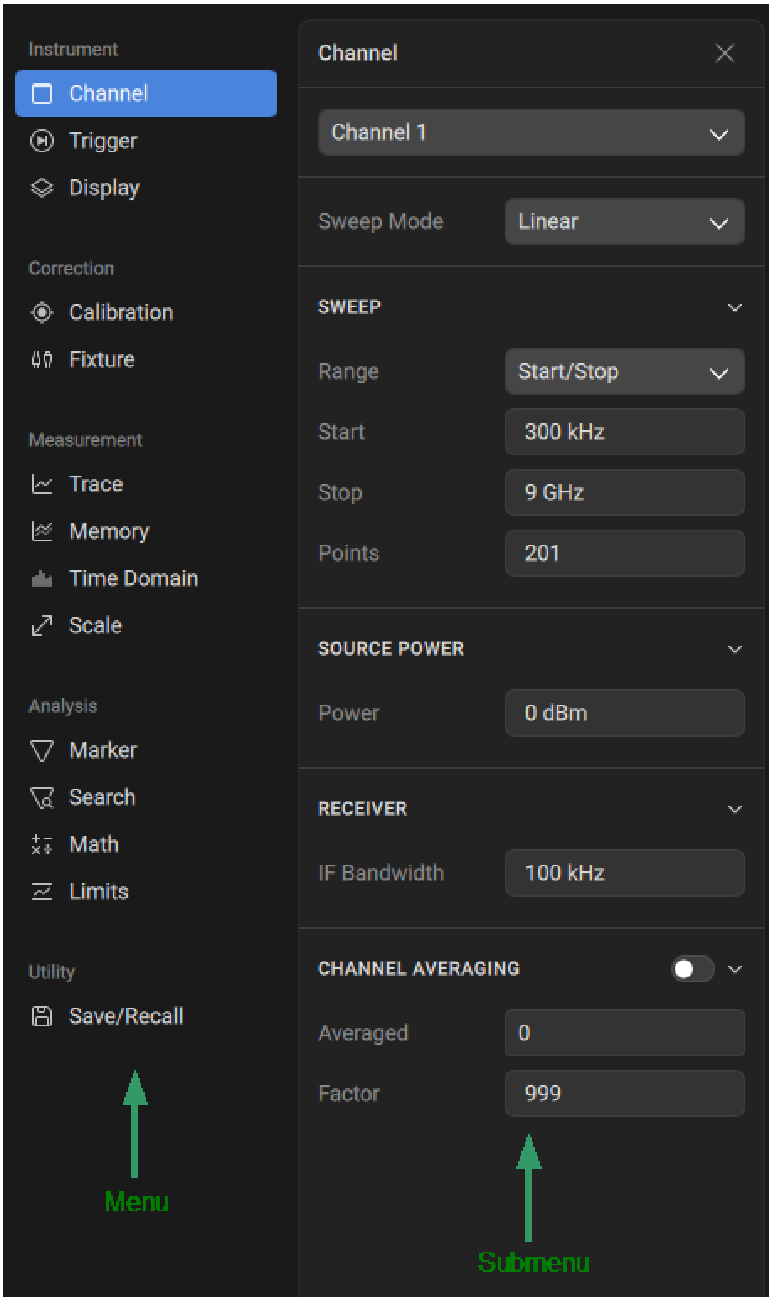
Button	Description
	Click on the button to add channel (corresponds to the **Add** button in Channel submenu, see Channel Allocation).
	Click on the button to remove active channel (corresponds to the **Delete** button in Channel submenu, see Channel Allocation).
	Click on the button to add trace (corresponds to the **Add Trace** button in Trace submenu).
	Click on the button to delete active trace (corresponds to the **Delete Trace** button in Trace submenu).
	Click on the button to add marker (corresponds to the **Add** button in Marker submenu, see Markers).
	Click on the button to delete the active marker (corresponds to the **Delete** button in Marker submenu, see Markers).
	Click on the button to define the trace scale automatically for all traces (corresponds to the **Autoscale All** button in Scale submenu, see Automatic Scaling).
	Click on the button to reset the Analyzer to the initial condition (corresponds to the **Preset** button in Save/Recall submenu, see Analyzer Presetting).

Button	Description
	Click on the button to save charts as image or docx, or to print charts.

Menu Overview

The sidebar consists of the following menus:

- Instrument
- Correction
- Measurement
- Analysis
- Utility



Instrument

Submenu	Functions
Channel	• Set active channel • Set sweep mode • Edit sweep (range, start, stop, and points) • Set source power • Set receiver • Turn ON/OFF channel averaging • Edit channel averaging parameters
Trigger	• Select active channel • Edit trigger parameters (mode, control) • Set trigger source • Set trigger scope • Set trigger input (port, event, polarity, position, and delay) • Set trigger output (port, type, function, and polarity)
Display	• Set active channel • Add or delete channel • Set active trace • Add or delete trace

Correction

Submenu	Functions
Calibration	• Select active channel • Open Calibration Wizard • Edit Calibration Kits • Edit Connectors • Edit calibration settings • Edit system impedance

Submenu	Functions
Fixture	• Open Fixture Wizard

Measurement

Submenu	Functions
Trace	• Set active channel and trace • Add or delete trace • Set trace measurement parameters • Set trace smoothing • Set trace hold • Change trace colors
Memory	• Set active channel and trace • Add to trace memory or delete from trace memory • Set math operations • Turn ON/OFF memory buffer • Set active memory
Scale	• Set active channel and trace • Set scaling • Set electrical delay • Set phase offset

Analysis

Submenu	Functions
Marker	• Set active channel and trace • Set display settings (active, stimulus, add/delete marker, reference marker ON/OFF) • Set marker properties
Search	• Set active channel and trace • Set active marker • Set search parameters

Submenu	Functions
	• Set search range • Set search properties
Math	• Set active trace • Set bandwidth/notch search • Set trace statistics • Set RF filter statistics
Limits	• Set active channel and trace • Set limit test parameters • Set ripple limit parameters • Set peak limit parameters

Utility

Submenu	Functions
Save/Recall	• Access user states • Preset state configuration • Set state configuration start • Save/recall touchstone data

Settings

Submenu	Functions
VNA Devices	• Disconnect device • Add device • Detected device list • Favorite device list • Demo list
Appearance	• Display • Diagram • Panel settings

Submenu	Functions
Measurement Units	• Distance units • Temperature • General decimal places • Marker decimal places
Remote Control	• Socket server • Port

Channel Windows

The channel window displays the measurement results in the form of traces and numerical values. Each window corresponds to one channel. The Analyzer hardware processes channels sequentially.

In turn, each channel window can display up to 64 traces of measured parameters. If there is more than one trace in a channel window, the way they are displayed can be changed in the diagram.

The general view of the channel window is represented in the figure below.

Channel Window

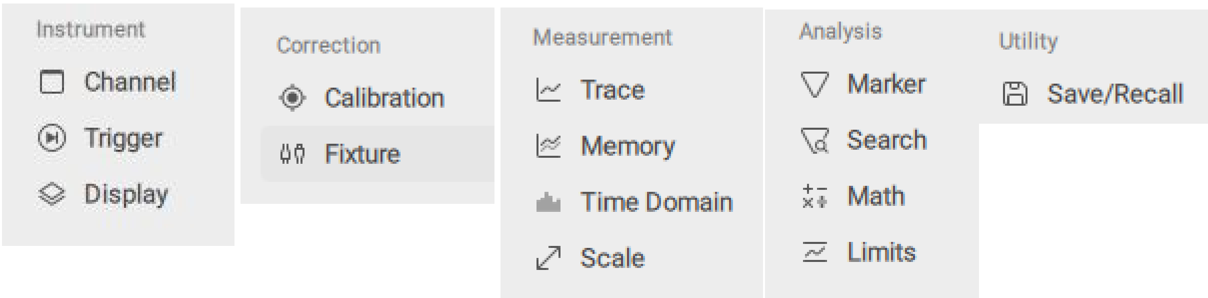
Each channel window contains a [Channel title](#) to be defined by the user, a [Trace manager](#) for displaying and changing to the name and parameters of the traces, a [Diagram](#) for displaying traces, as well as information about the channel status in the form of the [Channel Status Bar](#). Use the [Markers](#) feature to display the measurement values at the indicated trace points.

Each window has the following parameters:

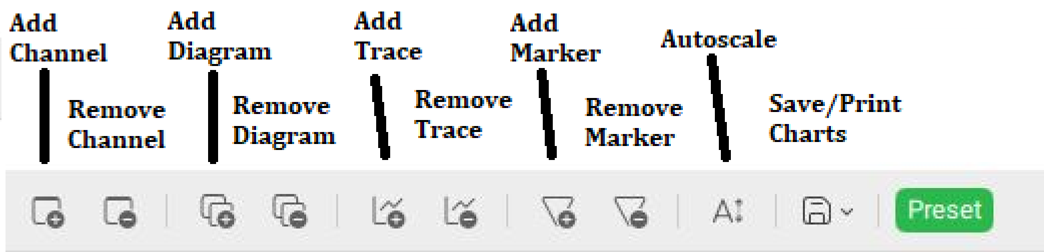
- Stimulus signal settings:
  - Frequency range
  - Number of Points
  - Sweep Type
  - Power level
- IF Bandwidth and average
- Calibration

Channel Title Bar

The menu is divided into sensible sections: Instrument, Correction, Measurement, Analysis, and Utility]. Frequently used functions, such as adding new traces, markers or diagrams, are implemented as buttons along the top of the UI.



Menu Selections

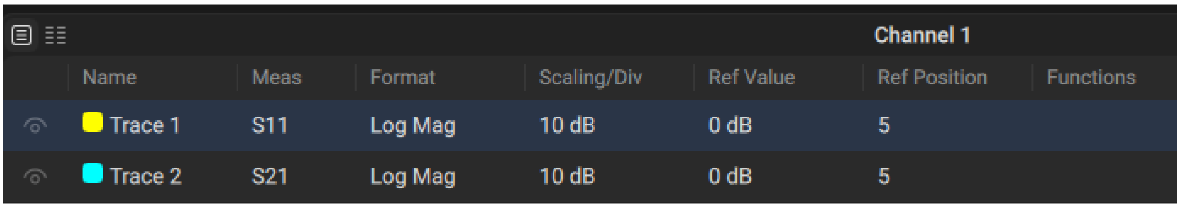


Top Buttons

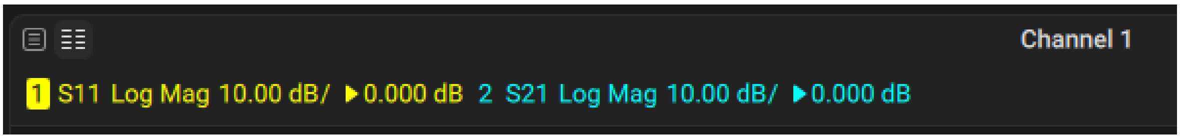
When a selection is made from the main menu panel (located on the left side of the screen) a corresponding sub-panel opens to the right. This sub-panel displays all available options and detailed settings related to the selected operation.

Trace Manager

The trace manager displays the name and parameters of a trace. There are three ways to present the trace manager: tabular and inline (see figure below). By default, the trace manager is displayed in the channel window as tabular.



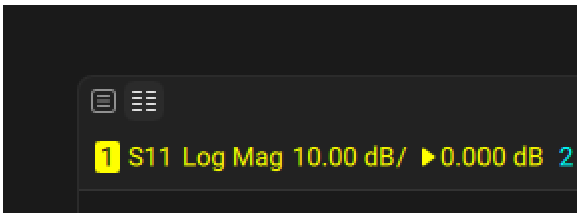
Tabular View



Inline View

Trace Manager

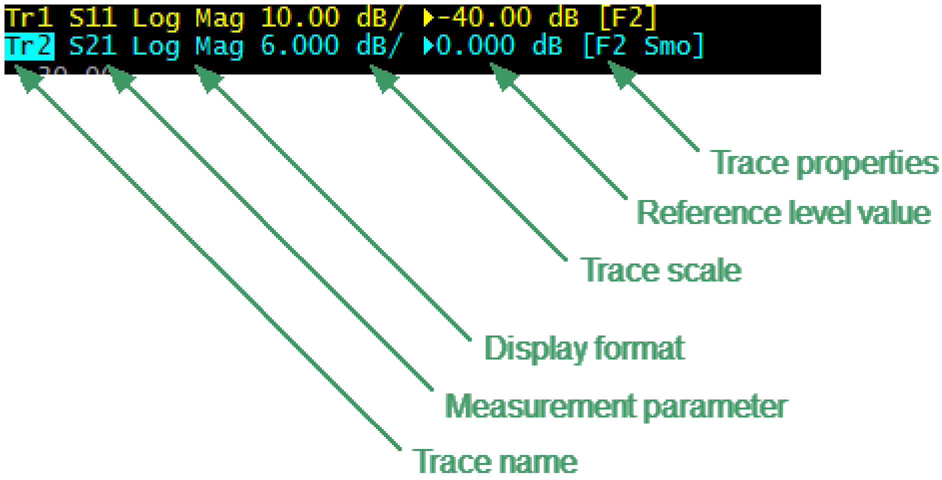
To switch between trace manager views, click on the required icon in the upper left corner of the channel window (see figure below). The left button is for tabular view and the right button is for inline view.



Buttons for Switching of Trace Manager View

NOTE	With a large number of traces, use the tabular or inline view of trace manager to reduce the clutter of the channel window.

The number of rows in the trace manager depends on the number of traces in the channel. The trace manager is represented in the figure below.

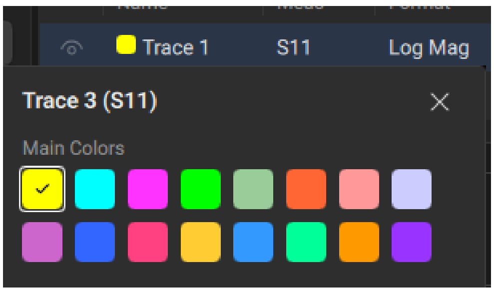


Trace Status Field (Tabular View)

The trace manager allows the user to set the channel parameters quickly and easily. The operations described in this section help to adjust the most frequently used settings. The complete set of channel functions can be accessed via the sidebar.

Each line contains the data of one trace of the channel:

- Displaying a trace in a channel, where  indicates that the trace is visible, and  indicates that the trace is hidden. The field is only available in tabular view.
- The color picker of data trace. The field is only available in tabular view. The color picker can be set using the trace manager.



To change the color of the trace, click on the colored square next to the trace name and select the desired color.

- The trace name from **1** to **64** (inline view), or the trace full name (tabular view). The active trace name is highlighted in an inverted color. For a detailed description of this setting, see Changing Trace Full Name.
- Measured parameter: **S11**, **S21**, ... **S1616**, or absolute power value: **T1(n)**, **T2(n)**, ..., **T16(n)**, **R1(n)**, **R2(n)**, ... **R16(n)**, or absolute power meter value: **P1**, **P2**,... **P16**. The conversion method is also displayed when measuring S-

parameters (see table below). For a detailed description of the setting, see S-Parameters and Conversion Function.0

Status	Symbol	Definition
Conversion	**Zr**	Reflection impedance
	Zt	Transmission impedance
	Yr	Reflection admittance
	Yt	Transmission admittance
	1/S	S-parameter inversion
	Ztsh	Transmission-shunt impedance
	Ytsh	Transmission-shunt admittance
	Conj	Conjugation

- Display format, e.g. **Log Mag**. For a detailed description of the setting, see Format Setting.
- Trace scale in measurement units per scale division, e.g. **10.0 dB**. For a detailed description of the setting, see Rectangular Scale and Circular Scale.
- Reference level value, e.g. **0.00 dB**. For a detailed description of the setting, see Rectangular Scale.
- Reference level position (the parameter is only available in tabular view), e.g. **5**. For a detailed description of the setting, see Rectangular Scale.
- Trace functions (see table below). Trace functions is indicated as symbols in square brackets in line view of the trace manager.

Function	Symbol	Definition
Error Correction	**RO**	OPEN response calibration
	RS	SHORT response calibration
	RT	THRU response calibration
	OP	One-path N-port calibration

Function	Symbol	Definition
	F1	Full one-port SOL calibration
	F2, F3, F4, F5, F6, F7, F8, F9, F10, F11, F12, F13, F14, F15, F16	Full N-port SOLT calibration (N take a value from 2 to 6/8/10/12/14/16. N depending on the Analyzer model, see corresponding datasheet)
Other Calibrations	**RC**	Receiver calibration
	PC	Power calibration
Data Analysis	**Z0**	Port impedance conversion
	Dmb	Fixture de-embedding
	Emb	Fixture embedding
	Pxt	Port extension
Trace Display	No indication	Data trace memory does not exist
	Data & Memory	Data and memory traces
	Data	Data trace only, memory exists
	Memory	Memory trace
	Off	Data and memory traces OFF
Trace Hold	No indication	Trace hold OFF
	Max	Hold the maximum value
	Min	Hold the minimum value

Function	Symbol	Definition
Memory Buffer	No indication	Memory buffer OFF (One memory trace)
	FIFO: n/m	Memory trace n (m is queue depth)

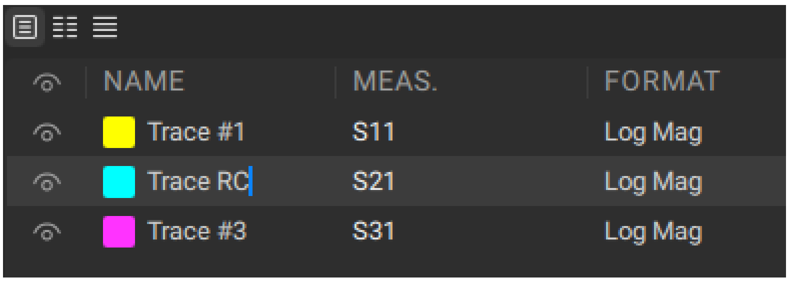
Function	Symbol	Definition
Math Operations	No indication	Data trace memory does not exist
	Data/Mem	Data / Memory
	Data*Mem	Data * Memory
	Data+Mem	Data + Memory
	Data–Mem	Data – Memory
	Off	Math operation OFF
Electrical Delay	**Del**	Electrical delay other than zero
Smoothing	**Sm**	Trace smoothing
Gating	**Gat**	Time domain gating

Changing Trace Full Name Trace Manager

1. Switch trace manager in tabular view.

2. Click on the trace name field in the trace manager.

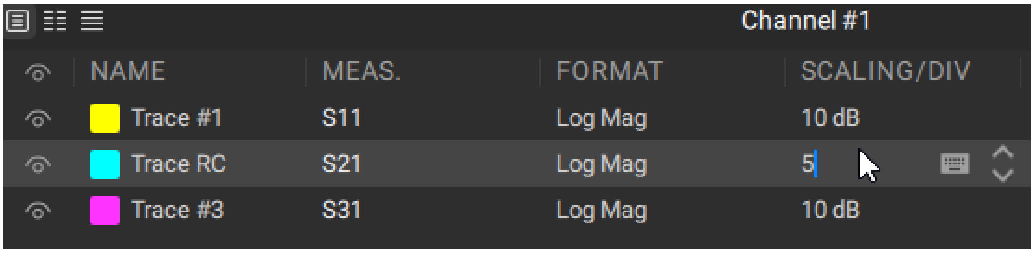
3. Enter a unique name of trace in the field.



Quick Trace Parameter Editing

1. Click on the required field in the trace manager.

2. Enter or select the required parameter value.

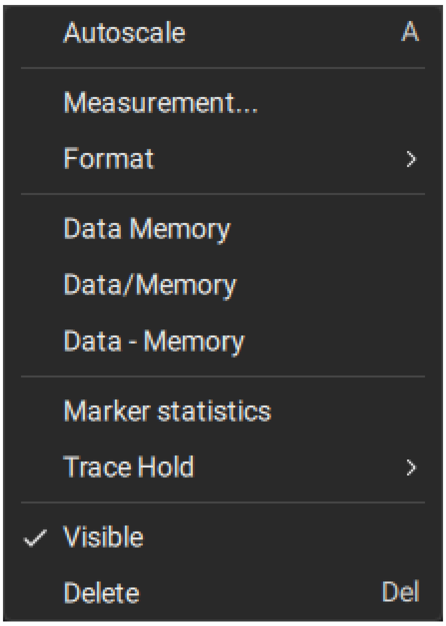


Quick Parameter Editing of Several Traces

Trace manager in tabular view allows the user to change parameters of several traces simultaneously. For example, you can assign the same format to all channel traces, or add memory traces to all traces at the same time.

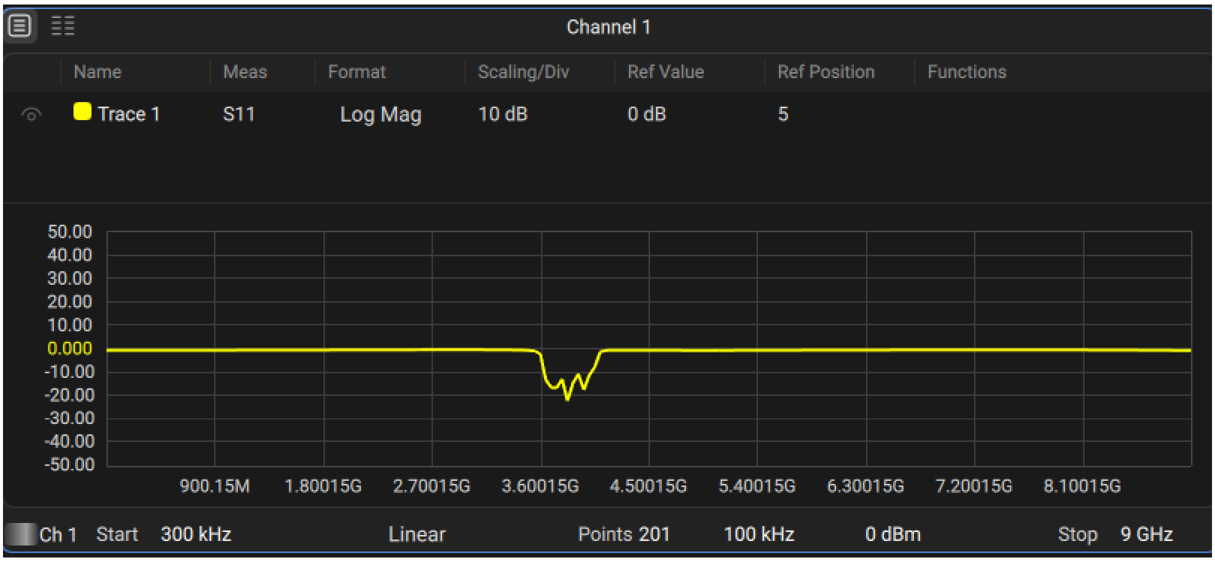
1. Click on the **CTRL** button on the keyboard and highlight the rows in the table.

2. Right-click on the selected rows and, in the pop-up window that opens, select the parameter to change (see figure below).



Diagram

The graph area in the channel window is called a diagram. The diagram displays traces and numeric data.



Diagram

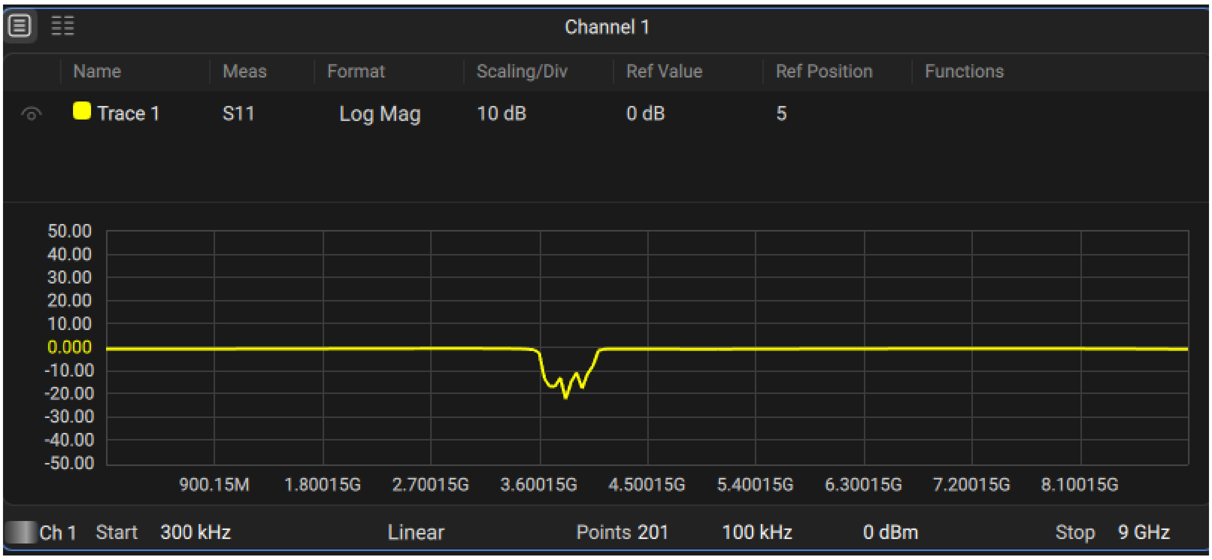
The diagram contains the following elements:

- **Vertical graticule label** displays the vertical axis numeric data for the active trace. The data for all traces can be displayed or hidden to gain more screen space for the trace display.
- **Horizontal graticule label** displays stimulus axis numeric data (frequency, power level or time). The horizontal graticule label can be hidden to gain more screen space for the trace display.
- **Reference level position** indicates the reference level position of the trace.
- **Markers** indicates the measured values at points along the active trace. The markers for all traces can be simultaneously displayed.
- **Marker functions**: statistics, bandwidth, flatness, RF filter.
- **Trace number** allows trace identification when printing in black and white.
- **Current stimulus position** indicator appears when sweep duration exceeds 1.5 sec.

NOTE	Using the diagram elements, the trace parameters can be easily modified using the mouse (as described in Quick Setting Using a Mouse).

Trace Layout in the Channel Window

If the number of the displayed traces is more than one, the traces can be rearranged. All the traces can be allocated to one diagram or each trace can be displayed on an individual diagram (See figures below).

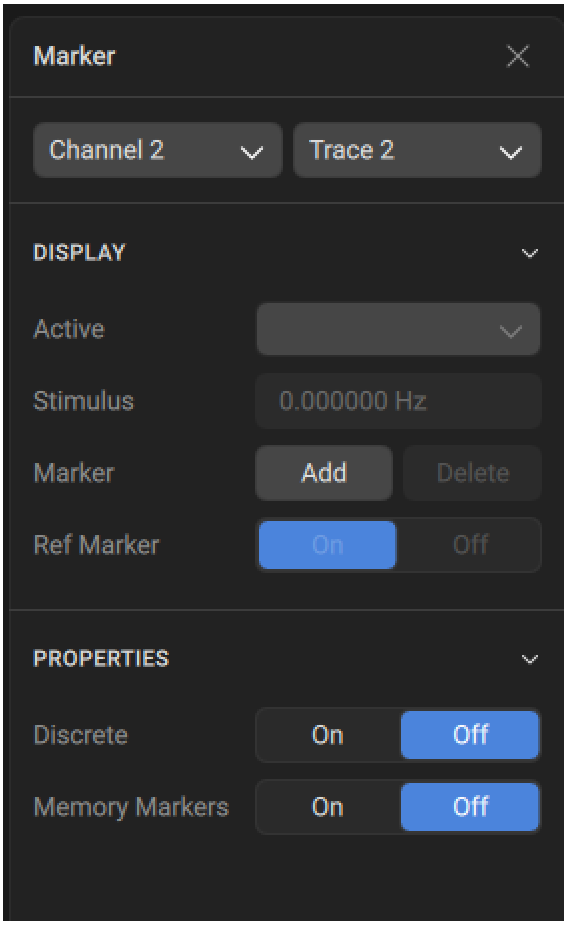


All traces in one diagram (example)

Each trace on an individual diagram (example)

Markers

The markers indicate the stimulus values and the measured values at selected points of the trace (See figure below).



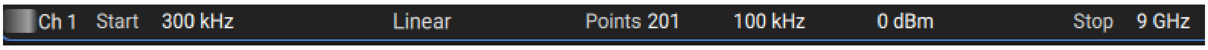
The markers are numbered from 1 to 15. The reference marker is indicated with an R symbol. The active marker is indicated in the following manners:

- Its number is highlighted with inverse color.
- The indicator on the trace is located above the trace.
- The stimulus indicator is fully colored.

NOTE	The use of markers is described in the Markers.

Channel Status Bar

The channel status bar is located in the bottom part of the channel window (See figure below).



Channel status bar

It contains the following elements:

- **Sweep progress** field displays a progress bar when the channel data is being updated.
- **Error correction** field displays the integrated status of error correction for S-parameter traces. The values of this field are represented in the table below. For a detailed description, see Error Correction Status.

Symbol	Definition	Note
Cor	Error correction is enabled. The stimulus settings are the same for the measurement and the calibration.	If the function is active for all traces — black characters on a gray background. If the function is active only for some of the traces (other traces are not calibrated) — white characters on a red background.
C?	Error correction is enabled. The stimulus settings are not the same for the measurement and the calibration. Interpolation is applied.	
C!	Error correction is enabled. The stimulus settings are not the same for the measurement and the calibration. Extrapolation is applied.	
Off	Error correction is turned off.	For all traces. White characters on a red background.
—	No calibration data. No calibration was performed.	

<table>
<tr><th>Symbol</th><th>Definition</th><th>Note</th></tr>
<tr><td>RC</td><td>Receiver correction is enabled. The stimulus settings are the same for the measurement and the calibration.</td><td rowspan="3">If the function is active for all the traces — black characters on a gray background.

If the function is active only for some of the traces (other traces are not calibrated) — white characters on a red background.</td></tr>
<tr><td>RC?</td><td>Receiver correction is enabled. The stimulus settings are not the same for the measurement and the calibration. Interpolation is applied.</td></tr>
<tr><td>RC!</td><td>Receiver correction is enabled. The stimulus settings are not the same for the measurement and the calibration. Extrapolation is applied.</td></tr>
</table>

Symbol	Definition	Note
PC	Power correction is enabled. The stimulus settings are the same for the measurement and the calibration.	If the function is active for all traces — black characters on a gray background. If the function is active only for some of the traces (other traces are not calibrated) — white characters on a red background.
PC?	Power correction is enabled. The stimulus settings are not the same for the measurement and the calibration. Interpolation is applied.	
PC!	Power correction is enabled. The stimulus settings are not the same for the measurement and the calibration. Extrapolation is applied.	

- **Port extension** field indicates whether this function is active and applied to one or all traces. If the function is enabled for all traces, black characters will be displayed on a gray background. If the function is enabled just for some of the traces, white characters will be displayed on a red background. For a detailed description, see Port Extension.
- **Fixture simulation** field indicates whether this function is active and applied to one or all traces. Fixture simulation includes the following operations: Z0 conversion, embedding, and de-embedding. If the function is enabled for all traces, black characters will be displayed on a gray background. If the function is enabled just for some of the traces, white characters will be displayed on a red background. For a detailed description, see Fixture Simulation.
- **Stimulus start** field allows for display and entry of the start frequency or power, depending on the sweep type. This field can be switched to indicate the stimulus center frequency, in this case the word «Start» will change to «Center». For a detailed description of stimulus setting, see Sweep Range.
- **Number of Points** field allows to display and entry the number of sweep points. The number of points can be set from 2 to the instrument maximum. For a detailed description, see Number of Points.
- **Sweep type** field allows for display and selection of the sweep type. The values of this field are represented in the table below. For a detailed description, see Sweep Type.

Symbol	Definition
Lin	Linear frequency sweep.
Log	Logarithmic frequency sweep.
Segm	Segment frequency sweep.
Pow	Power sweep.

- **IF bandwidth** field allows for display and setting of the IF bandwidth. The values can be set from the instrument minimum up to 1 MHz for some models. For a detailed description, see IF Bandwidth Setting.
- **Power level** field allows for display and entry of the port output power. In power sweep mode, the field switches to indicate the CW frequency of the source. For a detailed description, see Stimulus Power.
- **Averaging status** field displays the averaging status, if this function is enabled. The first number is the averaging current counter value, the second one is the averaging factor. For a detailed description, see Averaging Setting.
- **Stimulus stop** field allows for display and entry of the stop frequency or power, depending on the sweep type. This field can be switched to indication of stimulus span, in this case the word «Stop» will change to «Span». For a detailed description of stimulus setting, see Sweep Range.

Instrument Status Bar

The instrument status bar is located at the bottom of the screen.

Instrument status bar

Messages in the instrument status bar

Field Description	Message	Instrument Status	Note
RF signal	**RF out**	RF out ON or OFF.	
Instrument	**Instrument Name/Serial Number**	Displays the instrument name and serial number.	
Temperature	**temperature, degrees C**	Displays Analyzer temperature in degrees C.	
Cycle Time	**numeric value, ms**	Measured cycle time	

Setting Measurement Conditions

The section describes how to set the various measurement conditions of the Analyzer. To perform measurements, do the following according to each measurement task:

- Set the required number of the channels, and their parameters. Set the required number of traces and their parameters for each channel. For a detailed description, see Channel and Trace Setting.
- Set the stimulus signal parameters for each channel. For a detailed description, see Stimulus Settings.
- Assign the measured parameters, display format and scale for each trace. For a detailed description, see Measurement Parameters Settings, Format Setting, Scale Settings.
- If necessary, set the related trigger settings to synchronize measurements with any events. For a detailed description, see Trigger Settings.
- Set the optimization parameters to improve the signal to noise ratio. For a detailed description, see Measurement Optimization.

This section also describes how to quickly set the parameters of the analyzer using a mouse. For a detailed description, see Quick Setting Using a Mouse.

Channel and Trace Setting

The Analyzer supports 16 channels, each of which allows for measurements with stimulus parameter settings different from the other channels. The parameters related to a channel are listed in the table below.

Channel parameters

N	Parameter Description
1	Sweep Type
2	Sweep Range
3	Number of Points
4	Stimulus Power Level
6	CW Frequency
7	Segment Sweep Table
8	Trigger Mode
9	IF Bandwidth
10	Averaging
11	Calibration
12	Fixture Simulator

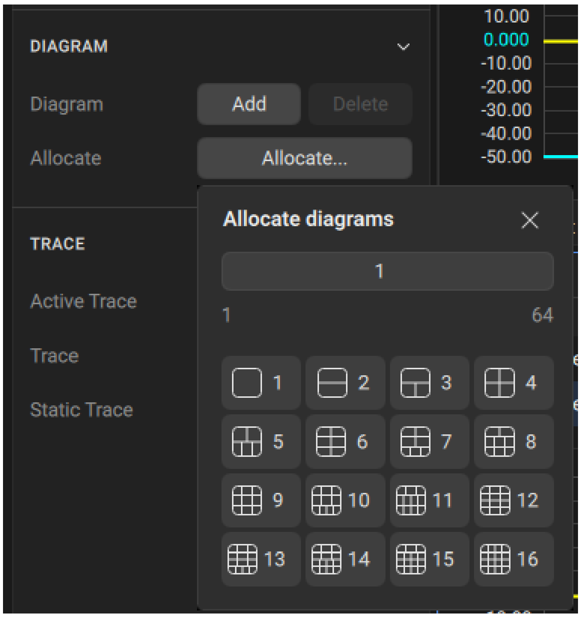
Each channel window can contain up to 16 different traces. Each trace is assigned a measured parameter (S-parameter), display format, and other parameters. The parameters related to a trace are listed in the table below.

Trace parameters

N	Parameter Description
1	Measured Parameter
2	Display Format
3	Scale Settings
4	Electrical Delay, Phase Offset
5	Memory Trace, Math Operation
6	Smoothing
7	Markers
8	Time Domain
9	Parameter Transformation
10	Limit Test
11	Ripple Limit Test
12	Peak Limit Test

Channel Allocation

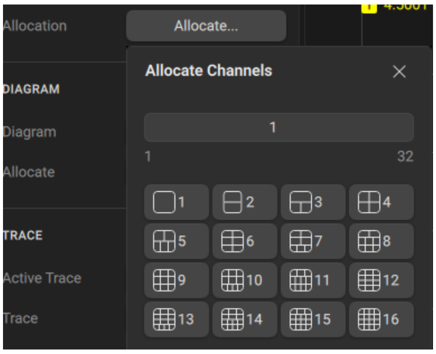
A channel is represented on the screen as an individual channel window. The screen can display many channel windows simultaneously. By default, one channel window opens.



In accordance with the layouts, the channel windows do not overlap each other. The channels open sequentially from low to high.

NOTE	For each open channel window, set the stimulus parameters, adjust other settings, and perform calibration. For a detailed description, see Stimulus Settings. Before changing a channel parameter setting or performing calibration of a channel, ensure that the channel is selected as active. For a detailed description, see Selection of Active Trace/Channel.

The measurements are executed for open channel windows sequentially. Measurements for any hidden channel windows are not performed.



To set the channel window layout, use the following:

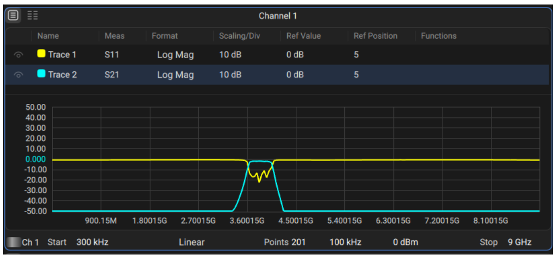
Display > Allocation

Then select the required number and layout of the channel windows in the menu.

DISP:SPL	Sets or reads out the number of the channel window layout on the screen.

Number of Traces

Traces can be displayed in one diagram, overlapping each other, or in separate diagrams within a channel window (See figures below).



Displaying two traces on the same diagram

Displaying two traces on two different diagrams

Add or remove a trace by pressing the Add Trace  or Remove Trace  buttons in the quick access toolbar.

CALC:PAR:COUN	Sets or reads out the number of traces in the channel.

All traces are assigned individual names, which cannot be changed. The trace name contains its number. The trace names are as follows: **Tr1**, **Tr2**, ... **Tr16**.

Each trace is assigned the following initial settings: measured parameter, format, scale, and color, which can be modified:

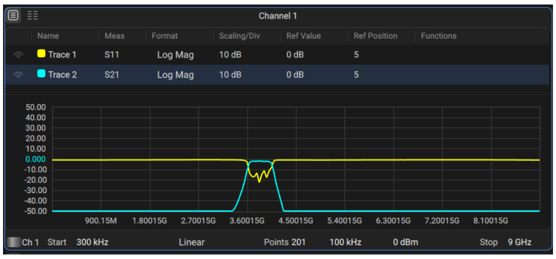
- By default, the measured parameters of the first two graphs are assigned the following values: **S11, S21**. When adding new traces, the assigned parameters will be cyclically repeated. For a detailed description of changing measured parameter see S Parameter.
- By default, the display format for all traces is set to logarithmic magnitude (dB). For a detailed description of changing display format see Format Setting.
- By default, the scale parameters are set as follows: division is set to 10 dB, reference level value is set to 0 dB, and the reference level position is in the middle of the diagram. For a detailed description of changing scale parameters see Scale Settings.

- The trace color is determined by its number. This color can be changed.

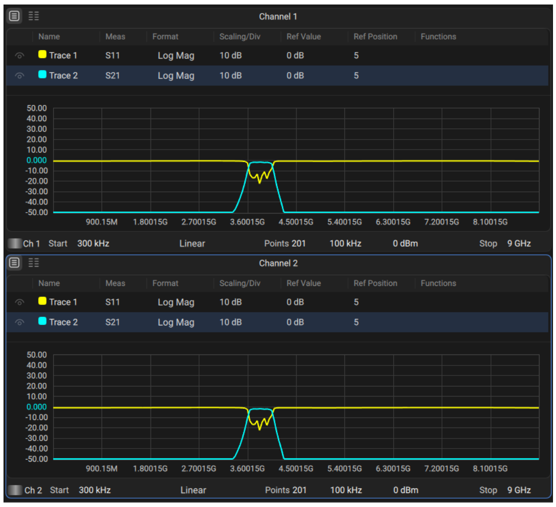
Trace Allocation

Each channel window can contain up to 64 different traces. Traces can be displayed in one diagram, overlapping each other, or in separate diagrams within a channel window (see figures below). Each trace can be placed into a separate diagram.

By default, the channel window contains one diagram with one trace.



Displaying Two Traces on the Same Diagram



Displaying Two Traces on Two Different Diagrams

Unlike channel windows, the number of traces and layout of the trace in diagrams are not related. The number of traces and the number of diagrams are set independently.

All traces are assigned an individual number which cannot be changed.

Each trace is assigned the following initial settings: name, measured parameter, format, scale, and color, which can be modified:

- By default, the display trace name is as follows: **Trace #n**, where **n** is a number of the trace.
- The measured parameters are set in the following succession: **S11, S21...S1616**. For a detailed description of changing measured parameter, see S Parameter.
- By default, the display format for all traces is set to logarithmic magnitude (dB). For a detailed description of changing display format, see Format Setting.
- By default, the scale parameters are set as follows: division is set to 10 dB, reference level value is set to 0 dB, and the reference level position is in the

middle of the diagram. For a detailed description of changing scale parameters, see Scale Settings.

- The trace color is determined by its number. This color can be changed. For a detailed description of changing color, see Color Picker of Trace.

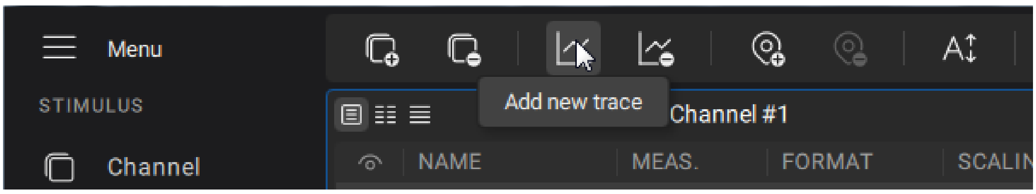
If two or more traces and diagrams need to be enabled, the number of traces and diagrams can be set as described below.

NOTE	The full cycle of the trace update depends on the set of measured S-parameters and the type of calibration. For example, a full cycle may include one sweep of port 1 as the signal source (when measuring S11). When measuring two traces S11, S22, two successive sweeps are performed. To measure the full matrix of S-parameters of the four-terminal network, four successive sweeps will be performed. When using a full 4-port calibration (SOLT), regardless of the number of traces and measured S-parameters, four successive sweeps are also performed. When using full n-port calibration (SOLT), regardless of the number of traces and measured S-parameters, n successive sweeps are also performed.

Adding Trace

1. Activate channel and diagram.

2. Press Add Trace in the Quick Access Toolbar.

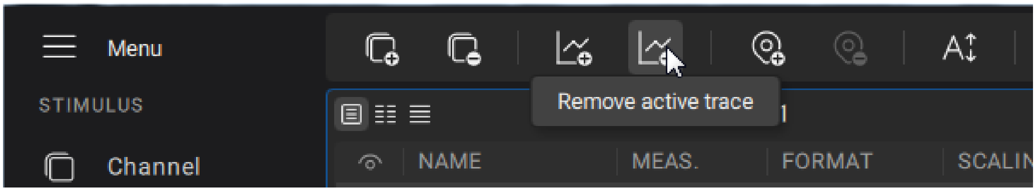


SCPI	CALCulate:PARameter:COUNt

Deleting Trace

1. Select the trace.

2. Press Remove Active Trace in the Quick Access Toolbar.



SCPI CALCulate:PARameter:COUNt

Adding Diagram

The new diagram is added last in the channel and contains one trace.

1. Select the channel.

2. Press Add Diagram in the Quick Access Toolbar.

Deleting Diagram

When a diagram is deleted, all its traces are transferred to the diagram of the first diagram.

1. Select the channel.

2. Press Remove Diagram in the Quick Access Toolbar.

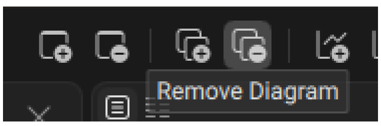
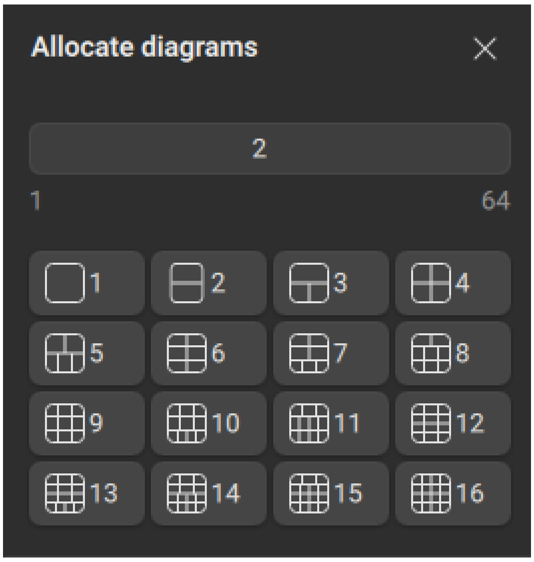


Diagram Allocation

The function places the required number of diagrams in the channel.

1	Click on the **Display** button in the sidebar. Press **Allocate** under the Diagram menu.
2	Enter the required number of diagrams in the textbox in **Diagram Allocation** pop-up window. If the number of diagrams does not exceed 16, you can choose one of the layouts below the textbox.



SCPI	DISPlay:WINDow:SPLit

Trace Visible

The function shows or hides a trace in the channel.

1 Setting trace visibility via the sidebar:

- Select the trace (see Selection of Active Trace, Diagram, Channel).
- Click on the **Trace** button in the main menu.
- Select or clear the **Trace visible** checkbox in the submenu.

Setting trace visibility via a trace manager:

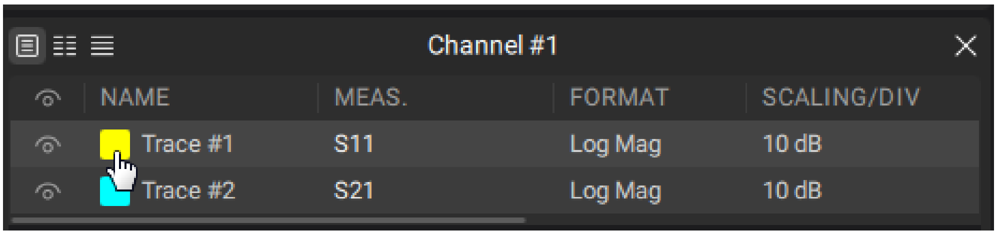
- Click on the icon  of the desired trace in the trace manager (tabular view only).

Color Picker of Trace

The color of traces can be customized if necessary.

Setting color via a trace manager:

- Click on the corresponding icon in the trace manager.



- Select the color of the data trace in the pop-up window using either a standard palette.

SCPI	DISPlay:COLor:TRACe:DATA

Selection of Active Trace/Channel

The selected control commands are applied to the active channel or trace respectively. If there are several diagrams in a channel, the active diagram must be selected.

The boundary line of the active channel window or active diagram is highlighted in blue. The active trace belongs to the active channel, and its title is highlighted in an inverse color.

Activate the channel/diagram/trace before setting the parameters of or deleting that channel/diagram/trace.

Selection of Active Channel

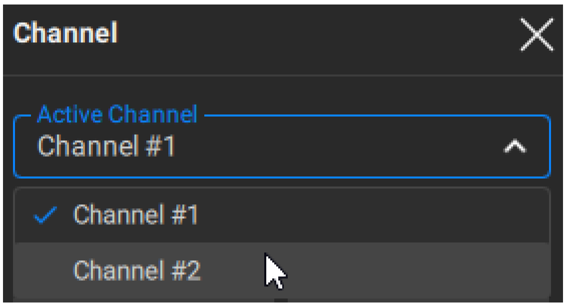
The active channel can be selected if the Analyzer window contains two or more channels.

The boundary line of the active channel window is highlighted in blue.

Active Channel

1 Click the **Channel** button in the main menu.

2 Click on the **Active channel** drop-down list in the submenu and select the required channel from the list.



SCPI	DISPlay:WINDow:ACTivate

NOTE	You can quickly activate the channel by clicking on it in the channel window.

Selection of Active Diagram

The active diagram can be selected if the active channel window contains two or more diagrams simply by clicking on the diagram in the channel window.

The boundary line of the active diagram is highlighted in blue.

1 Click on the desired diagram in the channel window.

NOTE	The diagram is activated automatically when the active trace is selected in it.

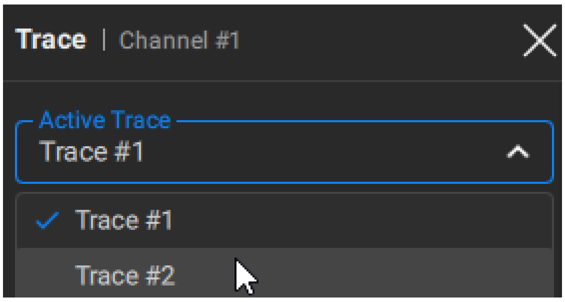
Selection of Active Trace

The active trace can be selected if the active channel window contains two or more traces.

The active trace is highlighted in a trace manager.

1. Click the **Trace** button in the main menu.

2. Click on the **Active Trace** drop-down list in the submenu and select the required trace from the list.



SCPI	CALCulate:PARameter:SELect

Stimulus Settings

This section describes how to set the stimulus signal parameters.

Stimulus — a signal with a known amplitude and phase, fed by the Analyzer to the device under test.

The stimulus parameter settings apply to each channel. Before setting the stimulus parameters of a channel the channel must be made active (See Selection of Active Trace/Channel).

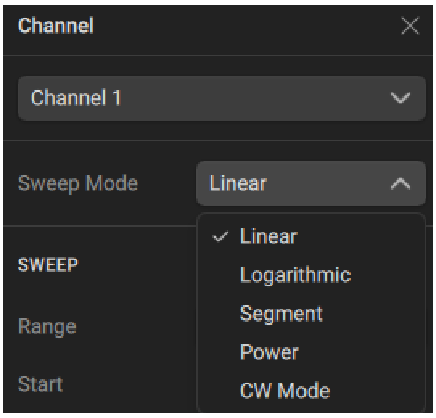
NOTE	To make maximize measurement accuracy, perform measurements with the same stimulus settings as were used for calibration.

Sweep Type

The sweep type determines how the stimulus range is scanned:

- By frequency (linear frequency sweeps, logarithmic frequency sweeps or segment sweep mode).
- By power (linear power sweep).

The channel to which the function is applied must be preselected as active (See Selection of Active Trace/Channel).



To set the sweep type:

Channel > Sweep Mode

Then select the sweep type:

- **Linear** — Linear frequency sweep.
- **Logarithmic** — Logarithmic frequency sweep.
- **Segment** — Segment frequency sweep.
- **Power** — Power sweep.
- **CW Mode** — Continuous wave sweep.

SENS:SWE:TYPE	Sets or reads out the sweep type.

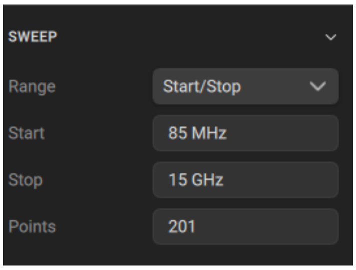
NOTE	The **Sweep Type** can be selected using the mouse (See Sweep Type Setting).

Sweep Range

The sweep range should be set for the linear and logarithmic frequency sweeps (Hz) and for the linear power sweep (dBm).

The sweep range can be set using either Start/Stop or Center/Span values.

The channel to which the function is applied must be preselected as active (See Selection of Active Trace/Channel).

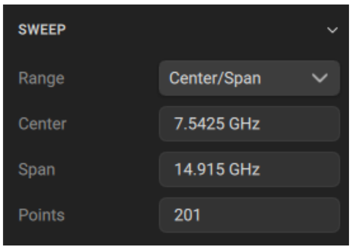


To enter the start and stop values of the sweep range:

Channel > Sweep > Start

Channel > Sweep > Stop

SENS:FREQ:STAR	Sets or reads out the stimulus start value of the sweep range for linear or logarithmic sweep type.
SENS:FREQ:STOP	Sets or reads out the stimulus stop value of the sweep range for linear or logarithmic sweep type.
SOUR:POW:STAR	Sets or reads out the power sweep start value when the power sweep type is active.
SOUR:POW:STOP	Sets or reads out the power sweep stop value when the power sweep type is active.



To enter center and span values of the sweep range:

Channel > Sweep > Center

Channel > Sweep > Span

SENS:FREQ:CENT	Sets or reads out the stimulus center value of the sweep range for linear or logarithmic sweep type.
SENS:FREQ:SPAN	Sets or reads out the stimulus span value of the sweep range for linear or logarithmic sweep type.

SOUR:POW:CENT	Sets or reads out the center value of the power sweep type.
SOUR:POW:SPAN	Sets or reads out the power span when the power sweep type is active.

NOTE	If power sweep is activated, the values on the **Start, Stop, Center** and **Span** softkeys will be represented in **dBm**.

NOTE	The **Start, Stop, Center** and **Span** values of the sweep range can be set using the mouse. Switch between **Start/Center** and **Stop/Span** modes with the mouse. The **Start/Center** and **Stop/Span** values can be set using the mouse.

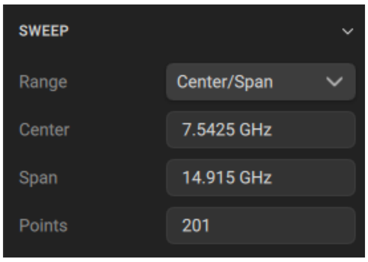
Number of Points

The number of points is the number of measurements gathered in a sweep cycle in the range of stimulus change.

The number of points should be set for the linear and logarithmic frequency sweeps, and for the linear power sweep.

Increase the number of points to get a larger trace resolution. To increase measurement performance, reduce the number of points to values that provide an acceptable trace resolution. To maintain high accuracy, the number of points in the calibration and in the actual measurements must be the same.

The channel to which the function is applied must be preselected as active (See Selection of Active Trace/Channel).



To enter the number of points:

Channel > Sweep > Points

SENS:SWE:POIN	Sets or reads out the number of measurement points.

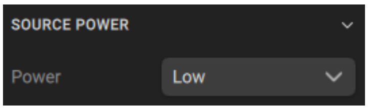
NOTE	The number of **Points** can be set using the mouse (See Number of Points Setting).

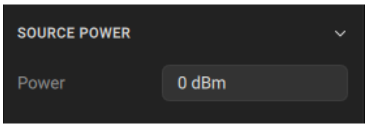
Stimulus Power

The stimulus power level should be set for the linear and logarithmic frequency sweeps.

For the segment sweep type, the method of power level setting described in this section can be used only if the same power level is set for all the segments of the sweep. For setting of individual power levels for each segment, see Segment Table Editing.

The channel to which the function is applied must be preselected as active (See Selection of Active Trace/Channel).





To enter the source power:

Channel > Source Power > Power

SOUR:POW	Sets or reads out the power level for the frequency sweep type.

NOTE	Setting the **Power** level is possible using the mouse (See Power Level/CW Frequency Setting).

CW Frequency

The CW frequency setting determines the fixed frequency for the linear power sweep.

The channel to which the function is applied must be preselected as active (See Selection of Active Trace/Channel).

To enter the CW frequency value, use the following softkeys:

Channel [CW Mode] > Frequency

SENS:FREQ	Sets or reads out the fixed frequency value when the power sweep type is selected.

NOTE	**CW frequency** can be set using the mouse (See Power Level/CW Frequency Setting).

RF Out Function

The RF Out function allows for temporary disabling of the stimulus signal. While the stimulus is disabled, measurements cannot be performed.

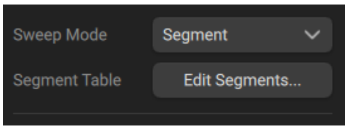
	To disable/enable stimulus, toggle the RF out toggle in the bottom left of the software ON/OFF.
OUTP	Turns the RF signal output ON/OFF.

NOTE	The **RF Out** function is applied to the Analyzer, not to individual channels. Indication of RF Out status appears in the instrument status bar (See Instrument Status Bar)

Segment Table Editing

The segment table determines the sweep parameters when segment sweep type is used (See Sweep Type).

The channel to which the function is applied must be preselected as active (See Selection of Active Trace/Channel).



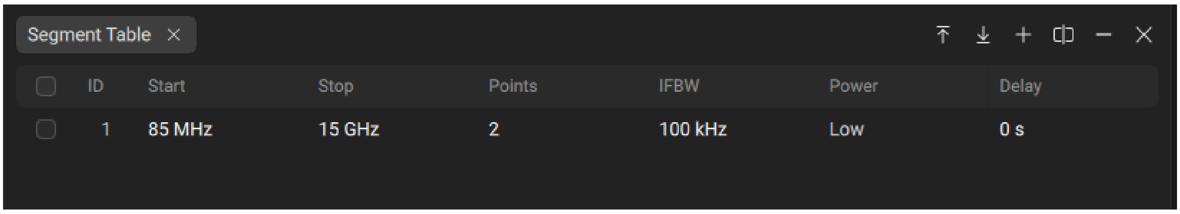
To open the segment table, use the following softkeys:

Channel > Sweep Mode > Segment

Edit Segments

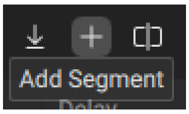
When editing the segments, the segment table will open in the lower part of the application. To hide the segment table, either click Edit Segments again or click Hide Segments.

The segment table layout is shown below (See figure below). The table has three mandatory columns: start frequency, stop frequency, and number of points, and three columns which can be optionally enabled/disabled: IF bandwidth, power level, and delay time.

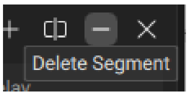


Segment Table Example

Each row describes one segment. The table can contain one or more rows. The number of segments is limited only by the instrument's maximum number of points.



To add a segment to the table, click the **Add** softkey. The new segment row will be entered below the highlighted one.

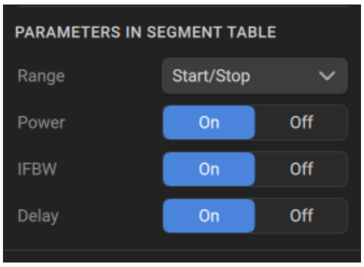


To delete a segment, click the **Delete** softkey. The highlighted segment will be deleted.

For any segment it is necessary to set the mandatory parameters: frequency range (start and stop) and number of points. The frequency range can be set either as Start / Stop, or as Center / Span.

To set the frequency range representation mode, use the Range dropdown to select between the **Start/Stop** and **Center/Span** options.

For any segment, the following additional parameter columns can be enabled: IF bandwidth, power level, and delay time. If such a column is disabled, the corresponding value set for linear sweep will be used (same for all the segments).



To enable the IF bandwidth column, toggle IFBW on.

To enable the power level column, toggle Power on.

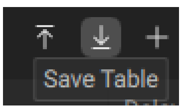
To enable the delay time column, toggle Delay on.

SENS:SEGM:DATA	Sets or reads out the array of the segment sweep table.

To set a parameter, click on its value field and enter the value. To navigate in the table, use the keys on the keyboard.

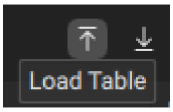
NOTE	Adjacent segments must not overlap in the frequency domain.

The segment table can be saved into *.SEG file to a hard disk and later recalled.



To save the segment table, click the **Save** softkey.

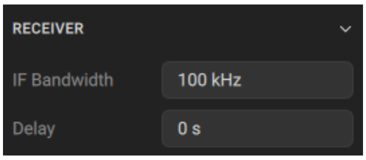
Then enter the file name in the appeared dialog.

	To recall the segment table, click **Load** softkey. Then select the file name in the appeared dialog.
MMEM:STOR:SEGM	Saves the segment table into a file.
MMEM:LOAD:SEGM	Recalls the segment table file. The file must be saved using the MMEM:STOR:SEGM command.

Measurement Delay

The measurement delay function allows for adding an additional time delay at each measurement point between the moment when the source output frequency becomes stable and the start of the measurement. This capability can be useful for measurements of electrically-long devices.

The channel to which the function is applied must be preselected as active (See Selection of Active Trace/Channel).

	To set the measurement delay time, use the following softkeys: **Channel > Receiver > Delay**
SENS:SWE:POIN:TIME	Sets or reads out the delay before measurement in each measurement point.

Trigger Settings

This section describes the trigger settings.

A trigger is a signal or event that starts the Analyzer measurement cycle. The measurement cycle, by default, includes the measurement of all opened channels. The Analyzer measures the channels sequentially one after another in one measurement cycle. At some conditions, the channel can be excluded from the measurement cycle (See Channel Initiation Mode and Trigger Scope).

For a detailed description of trigger state diagram see Trigger State Diagram.

The trigger settings include:

- Selection the trigger source (See Trigger Source).
- Selection the channel initiation mode (See Channel Initiation Mode).
- Setting the trigger scope (See Trigger Scope).

An external device can be used as a trigger source. For a detailed description of external trigger settings see External Trigger Settings.

The trigger output of the Analyzer can be a trigger source for other devices (See Trigger Output).

Trigger State Diagram

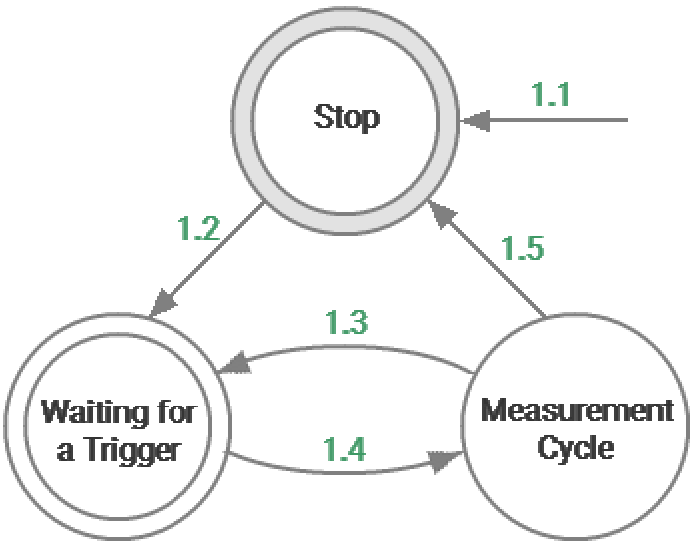
The trigger system operates at two levels: at the Analyzer level and at the channel level.

Analyzer States

The Analyzer can be in one of the following three states:

- **Stop** — the Analyzer waits for any channel to enter the **Initiated** state.
- **Waiting for a Trigger** — the Analyzer waits for the trigger signal. If the **Internal** trigger source (see Trigger Source) is selected, it is automatically generated.
- **Measurement Cycle** — all initiated channels are measured in turn.

The figure below shows the states of the Analyzer, and the transitions between them.



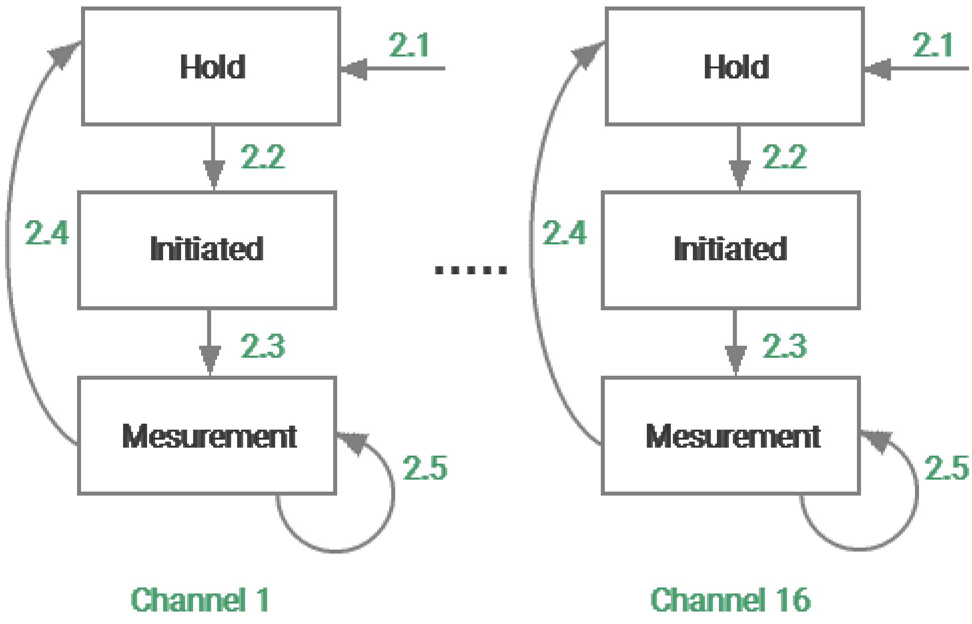
Analyzer states and transitions

Channel States

Channels can be in one of the three following states:

- **Hold** — the channel waits for the initiation. If the continuous initiation mode (see Channel Initiation Mode) is selected, the channel is automatically initiated.
- **Initiated** — the channel waits for the measurement after the trigger signal and measurement of other channels in the queue.
- **Measurement** — the channel is measured.

The figure below shows the channel states, and the transitions between them.



Channel states and transitions

The table below describes the transitions between analyzer and channel states.

Transition	Condition	Button	Command
1.1 To Stop	Power on	—	—
	Reset	**Preset**	SYST:PRESet, *RST
	Abort of the current measurement cycle.	**Trigger > Restart**	ABORt
	Changing Analyzer settings by user or by the SCPI command.	For example: **Stimulus > Start**	For example: SENS:FREQ:STARt
1.2	One or more channels make the transition 2.2	—	—

Transition	Condition	Button	Command
Stop –> Waiting for Trigger	to the Initiated state.		
1.3 Waiting for Trigger –> Measurement Cycle	Automatically, if the trigger source is set to **Internal**.	**Trigger source > Internal**	TRIG:SOUR INT
	At a signal arrival at the external trigger input, if the trigger source is set to **External**.	**Trigger source > External**	TRIG:SOUR EXT
	At a softkey pressing, if the trigger source is set to **Manual**.	**Trigger source > Manual** **Trigger > Trigger**	TRIG:SOUR MAN
	Upon receipt of SCPI command, if the trigger Source is set to **Bus**.	**Trigger source > Bus**	TRIG:SOUR BUS TRIG:SING, TRIG, *TRG
1.4 Measurement Cycle –> Waiting for Trigger	At the end of a measurement cycle, when at least one channel has the **Continuous** initiation mode.	**Trigger > Continuous**	INIT:CONT ON
	After measuring a point, when the **On Point** trigger function is active.	**Ext Trigger > Event > On Point**	TRIG:POIN ON

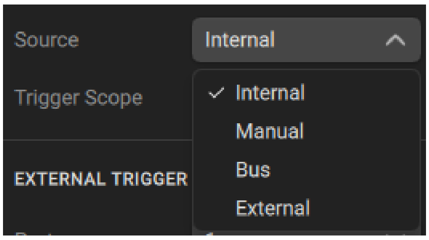
Transition	Condition	Button	Command
1.5 Measurement Cycle –> Stop	At the end of a measurement cycle, when the **Continuous** initiation mode is disabled for all channels.	**Trigger > Hold All Channels**	—
2.1 To Hold	The same condition as transition 1.1	—	—
	When the Initiation Mode of the channel has been set to **Hold**.	**Trigger > Hold**	INIT:CONT OFF
2.2 Hold –> Initiated	Every time if the **Continuous** initialization mode of the channel is turned on.	**Trigger > Continuous**	INIT:CONT ON
	Once when the **Single** initiation mode of the channel has been set.	**Trigger > Single**	INIT
2.3 Initiated –> Measurement	Upon the occurrence of one of the conditions transition 1.3 and after measurement of other channels in the queue.	—	—

Transition	Condition	Button	Command
2.4 Measurement –> Hold	At the end of channel measurement.	—	—
2.5 Repeat measurement	If the averaging trigger function is on, the measurement repeats N times, where N is averaging factor.	**Average > Avg Trigger > On**	TRIG:AVER

Trigger Source

One of four trigger sources can be selected. This setting works at the Analyzer level.

Trigger Source	Function
Internal [default]	The Analyzer generates a trigger signal automatically when needed.
External	A trigger signal is a logic signal at the external trigger input (See External Trigger Settings).
Bus	The trigger signal is generated by a command from the program controlling the Analyzer via SCPI.
Manual	The trigger signal is generated by pressing the **Trigger** softkey in the Analyzer software.



To set the trigger source, use the following softkeys:

Trigger > Source

Then select the required trigger source:

- **Internal**
- **External**
- **Manual**
- **Bus**

TRIG:SOUR	Selects the trigger source.

Channel Initiation Mode

The channel initiation mode determines whether the channel will be included in the measurement cycle when a trigger signal is detected.

The channel to which the function is applied must be preselected as active (See Selection of Active Trace/Channel).

Channel Initiation Mode	Function
Continuous [default]	The channel automatically transits to the Initiated state at the end of each measurement.
Single	The channel is initiated once. At the end of the measurement, the channel goes into the Hold state.
Hold	The channel is idle and not updating.

	To set the channel initiation mode, go to the Trigger menu. Then select the required channel initiation mode: • **Hold** • **Single** • **Continuous**
INIT:CONT	Turns the continuous initiation mode ON/OFF.
INIT	Sets the single initiation mode once.
	To set the appropriate mode for all channels, toggle **Apply to All Ch** ON/OFF.
INIT:CONT:ALL	Turns the continuous initiation mode for all channels ON/OFF.
	Restart softkey aborts the sweep and transits the Analyzer to stop state, then if there are channels in the continuous initiation state the Analyzer transits to the waiting for a trigger state (See Trigger State Diagram).

	Trigger > Trigger Control [Restart]
ABOR	Aborts the sweep.

Trigger Scope

The trigger scope function selects whether all initiated channels or an active channel, if initiated, will be measured on a trigger condition.

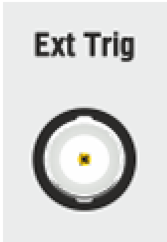
Trigger Scope	Function
All Channel [default]	All initiated[1] channels will be measured on a trigger condition.
Active Channel	Only the active channel, if initiated, will be measured on a trigger condition.
[1] For a detailed description of the channel initiation mode settings, see Channel Initiation Mode.	

	To set the trigger scope, go to: **Trigger > Trigger Scope [All Ch | Active Ch]**
TRIG:SCOP	Sets or reads out the trigger scope.

External Trigger Settings

This section describes settings of the external trigger.

The logic signal at the **Ext Trig** on the rear panel of the analyzer is an external trigger signal (See Instrument Series).



External Trigger Signal Connector

To work with an external trigger:

- Select trigger source **External** (See Trigger Source).
- Set the external trigger event, polarity, position and delay (See the subsections in this section).

External Trigger Event

This setting allows to select the external trigger event.

Trigger event	Function
On sweep [default]	One trigger signal starts a full measurement cycle, that is, the measurement of all frequency points of all channels included in the measurement cycle.
On point	One trigger signal starts the measurement of one frequency point of a channel. The next trigger signal starts the measurement of the next frequency point of the channel, and so on.

NOTE	If the **Averaging Trigger** function and the **On point** trigger function are enabled at the same time, the **On point** trigger function has priority. In this case, N * P trigger signals are required to complete the averaging, where N is the averaging factor, and P is the number of points.

To select an external trigger event, use the following softkeys:

Trigger > External Trigger [Event]

Then select the required external trigger event:

- **On Sweep**
- **On Point**

TRIG:POIN	Turns the point trigger feature ON/OFF.

External Trigger Polarity

Trigger polarity	Function
Negative Edge [default]	The negative edge of the input signal of an external trigger is a trigger signal.
Positive Edge	The positive edge of the input signal of an external trigger is a trigger signal.

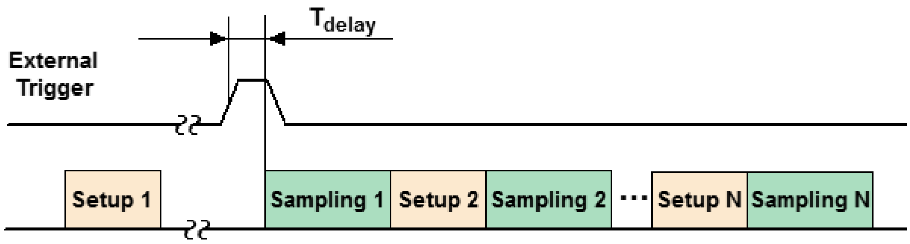
	To select external trigger polarity: **Trigger > External Trigger > Polarity** Then select the required external trigger polarity: • **Negative Edge** • **Positive Edge**
TRIG:EXT:SLOP	Sets or reads out the polarity of the external trigger.

External Trigger Position

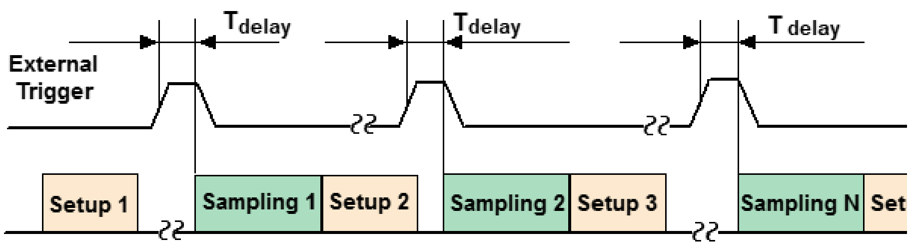
The position of the external trigger determines the moment when the Analyzer expects an external trigger signal — before the frequency setup or before measuring (ADC sampling). The frequency setup precedes the measurement for each frequency point.

<table>
<tr><th>Trigger Position</th><th>Function</th></tr>
<tr><td>Before Sampling

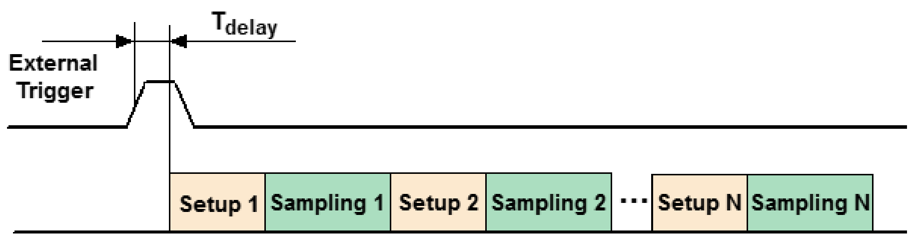
[default]</td><td>The trigger signal is expected before the ADC sampling when the frequency is already set. After sampling the Analyzer automatically transits to the next frequency (See figure below).

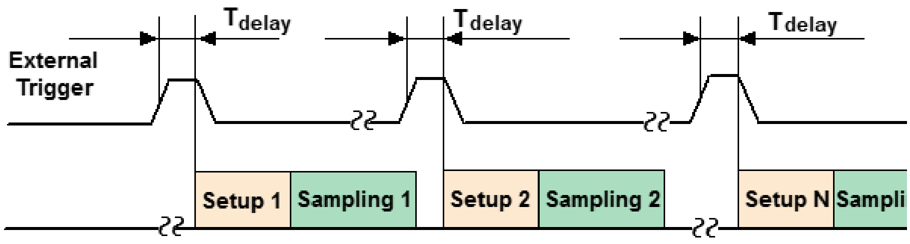


Before Sampling, Point trigger is OFF



Before Sampling, Point trigger is ON</td></tr>
<tr><td>Before Setup</td><td>The trigger signal is expected before the frequency setup. The frequency setup starts when the trigger signal arrives (See figure below). After the frequency setup is completed, the Analyzer begins ADC sampling.


</td></tr>
</table>

<table>
<tr><th>Trigger Position</th><th>Function</th></tr>
<tr><td></td><td>Before Setup, Point trigger is OFF


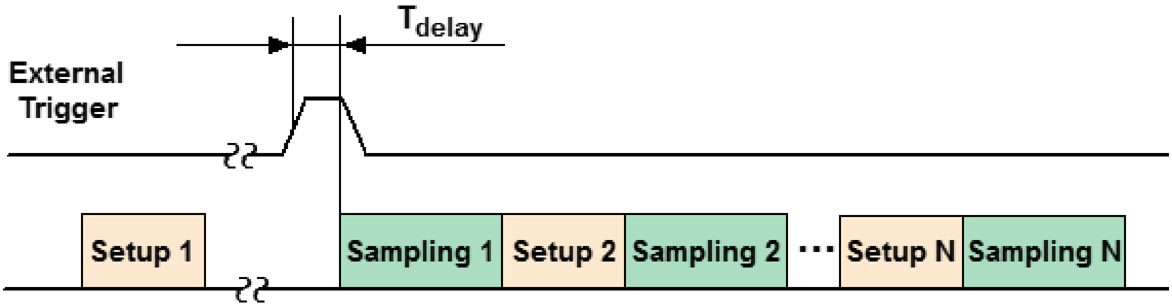
Before Setup, Point trigger is ON</td></tr>
</table>

NOTE	This function is intended for use in conjunction with the **On Point** trigger function. In case of the **On Sweep** trigger function, the trigger position will be performed only for the first sweep point.

	To select external trigger position: **Trigger > External Trigger > Position** Then select the required external trigger position: • **Before Sampling** • **Before Setup**
TRIG:EXT:POS	Selects the position of the external trigger.

External Trigger Delay

The external trigger delay sets the response delay with respect to the external trigger signal (See figure below).



External Trigger Delay

	To set the external trigger delay: **Trigger > External Trigger > Delay**
TRIG:EXT:DEL	Sets or reads out the response delay with respect to the external trigger signal.

Measurement Parameters Settings

This section describes the settings for the measurement parameter selection. The parameter selection applies to traces within a channel.

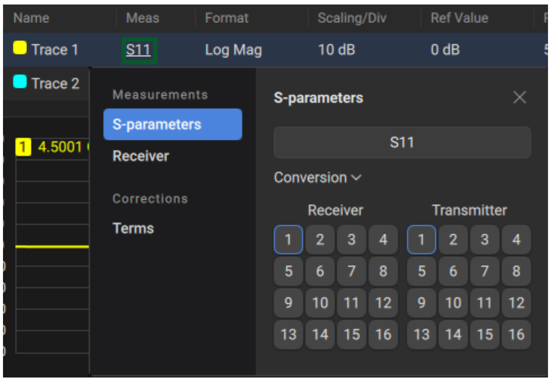
The Analyzers allows for:

- S-Parameter measurement (See S-Parameters)..

S-Parameters

A measured S-parameter(S11, S21) is set for each trace. The trace to which the function is applied must be preselected as active (See Selection of Active Trace/Channel).

For a detailed description of the principle of measuring S-parameters see in Principle of measuring S-parameters.



To set the measured parameter, click on the text under **Meas** in the trace manager, then set the desired parameter using the pop-up.

CALC:PAR:DEF	Selects the measurement parameter of the trace.

NOTE	Measured data can be set using the mouse (See Measured Data Setting).

Format Setting

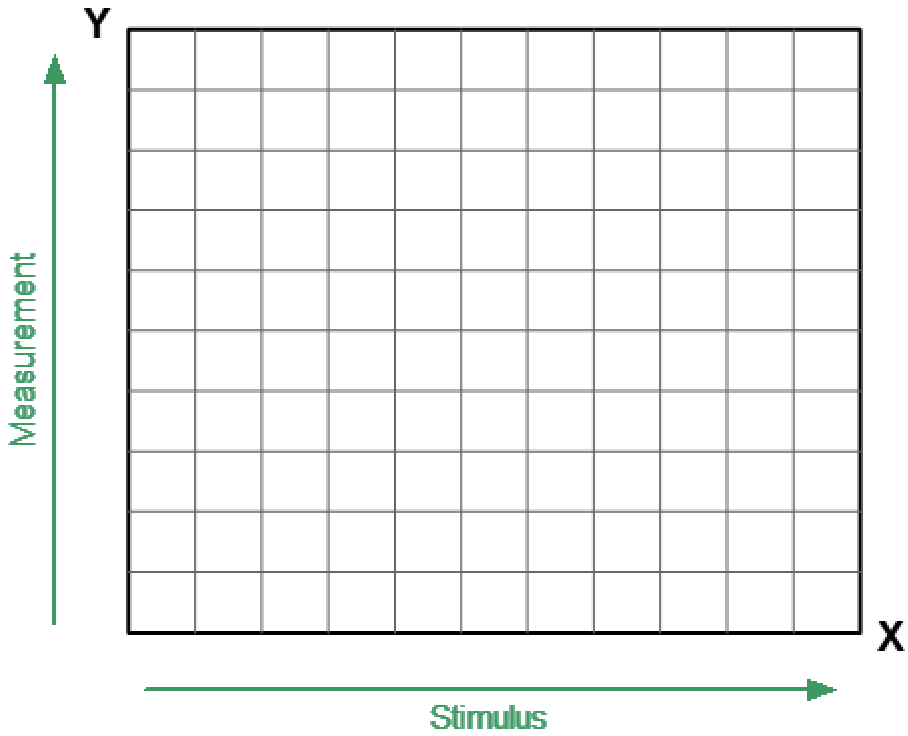
The format setting determines how measured data will be presented on the diagram.

The Analyzer offers three S-parameter measurement display types:

- Rectangular format
- Polar format
-

Rectangular Formats

In this format, stimulus values are plotted along X-axis and the measured data are plotted along Y-axis (See figure below).



Rectangular format

To display complex-valued S-parameters along the scalar Y-axis, it must be transformed into a real number. Rectangular formats involve various types of transformation of an S-parameter

$$S = a + j \cdot b,$$

where a — real part of S-parameter complex value,

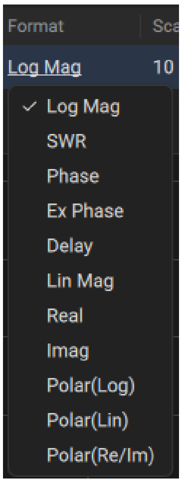
b — imaginary part of S-parameter complex value.

There are eight types of rectangular formats depending on the measured value plotted along Y-axis (See table below).

Rectangular Formats

Format Type Description	Label	Data Type (Y-axis)	Measurement Unit (Y-axis)
Logarithmic Magnitude	**Log Mag**	S-parameter logarithmic magnitude: $20 \cdot \log\|S\|$, $\|S\|=\sqrt{a^2+b^2}$	Decibel (dB)
Voltage Standing Wave Ratio	**SWR**	$\frac{1+\|S\|}{1-\|S\|}$	Dimensionless value
Phase	**Phase**	S-parameter phase from –180° to +180°: $\frac{180}{\pi} \cdot arctg\frac{b}{a}$	Degree (°)
Expanded Phase	**Expand Phase**	S-parameter phase, measurement range expanded to from below –180° to over +180°	Degree (°)
Group Delay	**Group Delay**	Signal propagation delay within the DUT: $-\frac{d\varphi}{d\omega}$, $\varphi = arctg\frac{b}{a}$, $\omega = 2\pi \cdot f$	Second (sec.)
Linear Magnitude	**Lin Mag**	S-parameter linear magnitude: $\sqrt{a^2+b^2}$	Dimensionless value
Real Part	**Real**	S-parameter real part: $a = re(S)$	Dimensionless value
Imaginary Part	**Imag**	S-parameter imaginary part: $b = im(S)$	Dimensionless value

The format for each trace of the channel can be selected individually. The trace must be activated before setting the format.



To choose a rectangular format, click on the text under Format in the trace manager.

Then select the desired format:

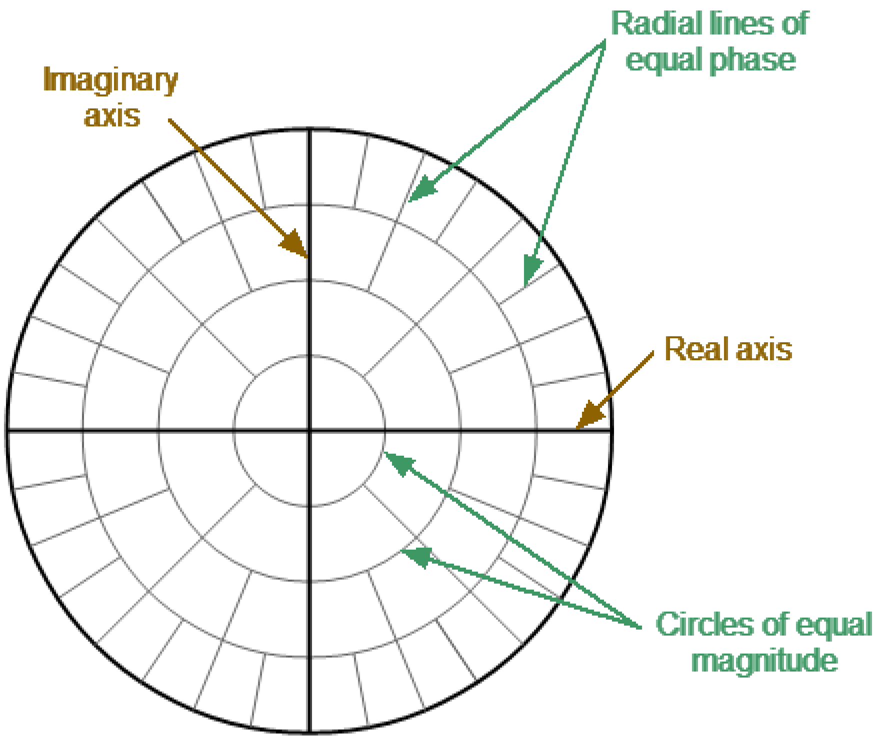
- **Logarithmic magnitude**
- **SWR**
- **Phase**
- **Expanded phase**
- **Group delay**
- **Linear magnitude**
- **Real part**
- **Imaginary part**

CALC:FORM	Sets or reads out the trace format.

NOTE	The display format can be set using the mouse (See Display Format Setting).

Polar Format

The Polar format is used to display the amplitude and phase of the reflection coefficient (Γ) when measuring S11 . The complex reflection coefficient values are displayed on the polar diagram in the complex plane. The complex plane is formed by the real horizontal and the imaginary vertical axes. The grid lines correspond to points of equal amplitude and phase (See figure below).

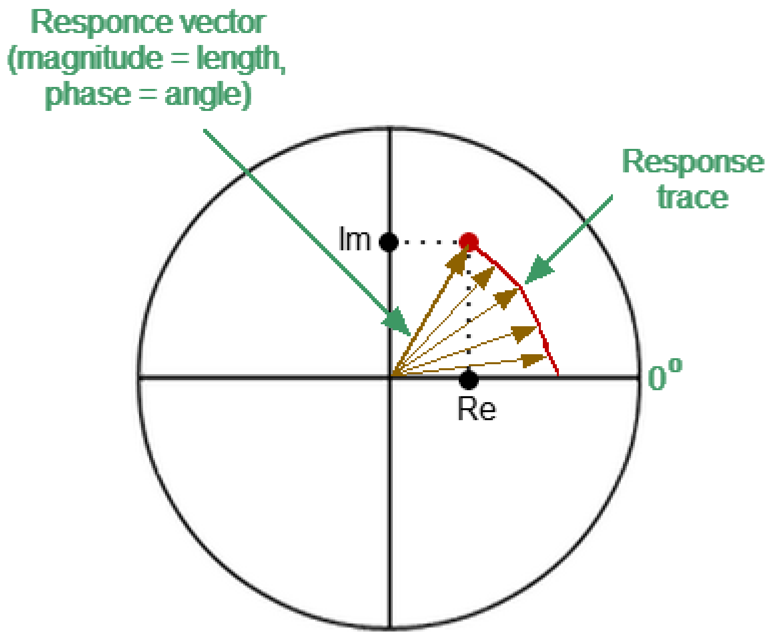


Polar format

NOTE

On circular diagrams (Polar and Smith chart), any point of the trace can be defined in the following two ways (See figure below):

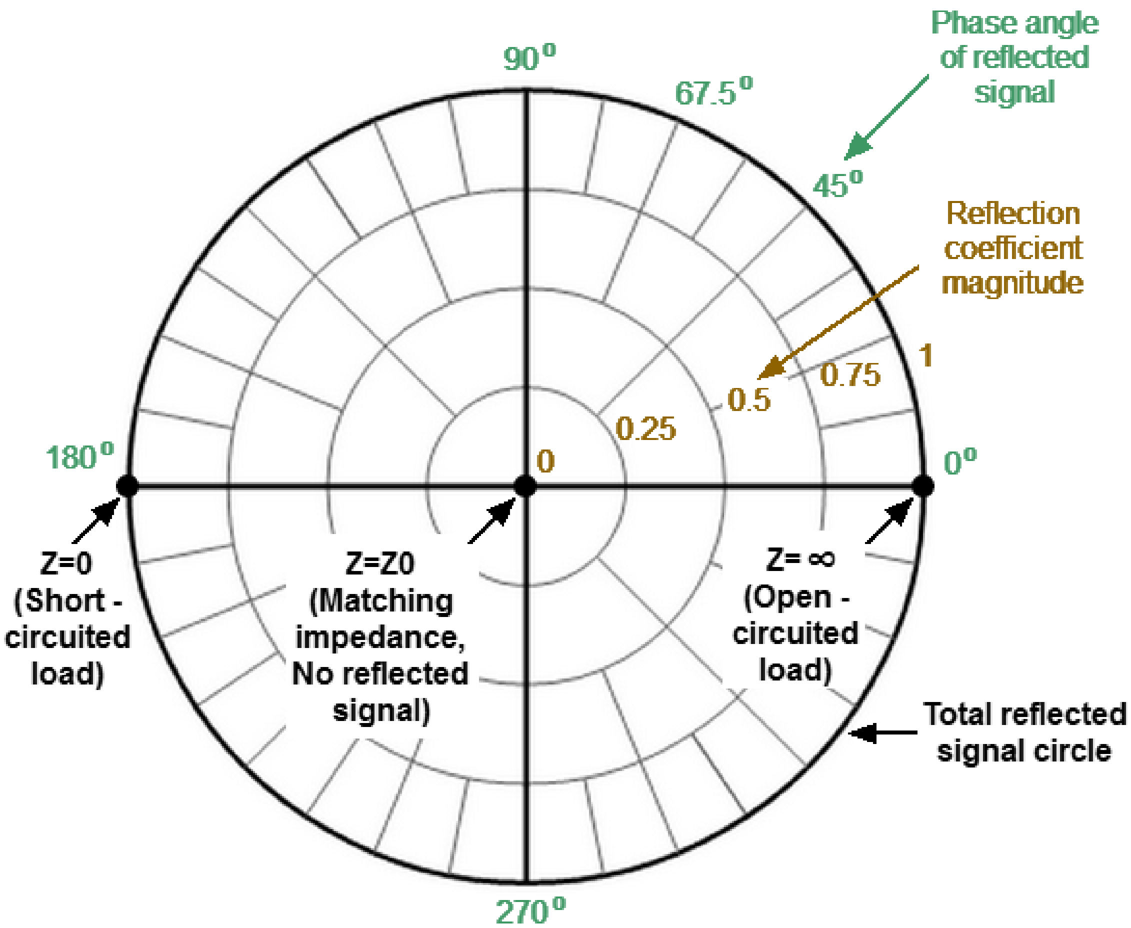
- Coordinates of the point (Re, Im) on the real and imaginary coordinate axes.
- Parameters of the vector directed to the point from the center of the diagram. The length of this vector is equal to the response amplitude, and the angle between the vector and the positive part of the real coordinate axis is equal to the phase of the response. The angle is calculated counterclockwise.



NOTE

Traces on all types of Smith chart and polar format are the same, the Analyzer replaces the base grid and default marker format when switching formats.

The Polar format diagram with the characteristic points is shown in the figure below.



Properties of Polar format

Basic properties of the Polar format:

- The center of the diagram corresponds to the reflection coefficient Γ = 0 (reference impedance Z0 on the input test port of the DUT when measuring S11

Scale Settings

The section describes how to set the scale for the different available formats.

The scale setting options depend on the selected data display format: rectangular format or circular format. For a detailed description of the scale settings for the different formats, see Rectangular Scale and Circular Scale (Polar and Smith).

It is possible to apply the Automatic Scaling function for both formats.

These functions are also available when using the rectangular format:

- Reference Level Automatic Selection

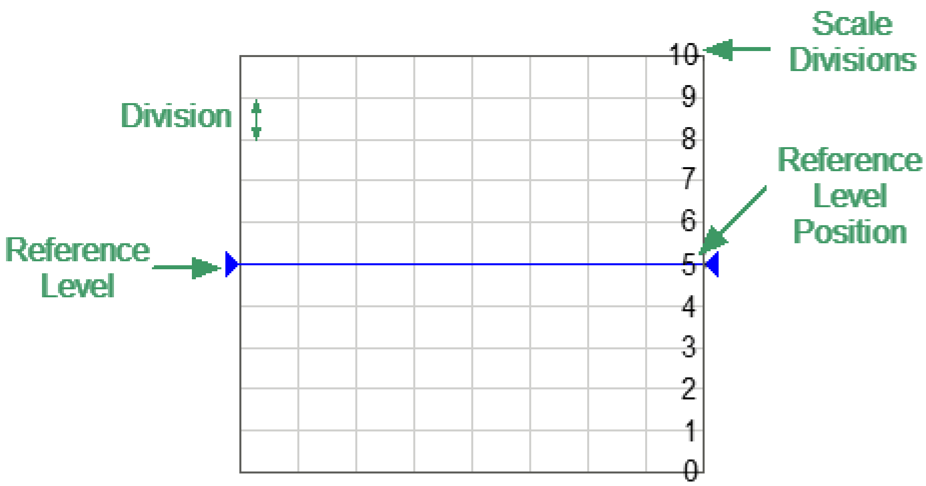
The scaling function is under trace settings.

This section also describes the electric delay setting functions (See Electrical Delay Setting) and phase offsets (See Phase Offset Setting).

Rectangular Scale

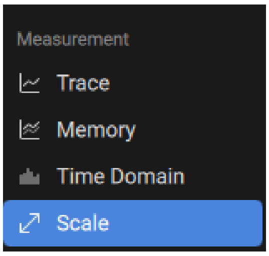
For rectangular format, the following parameters can be set (See figure below):

- scale division
- reference level value
- reference level position
- number of scale divisions



Rectangular scale

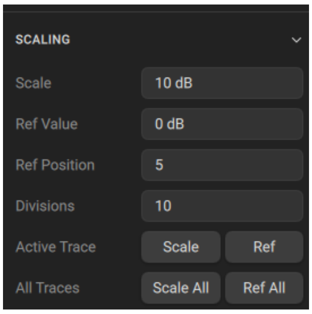
The scale of each trace can be set independently. The trace to which the function is applied must be preselected as active (See Selection of Active Trace/Channel).



To set the scale of a trace:

Scale > Scale

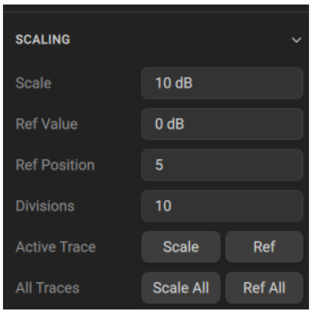
DISP:WIND:TRAC:Y:PDIV — Sets or reads out the trace scale. Sets the scale per division.



To set the reference level:

Scale > Ref Value

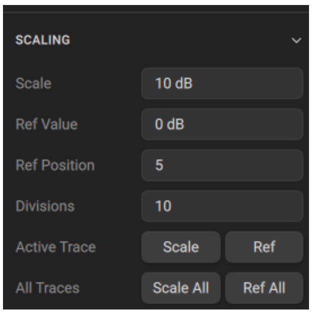
DISP:WIND:TRAC:Y:RLEV — Sets the value of the reference line (response value on the reference line).



To set the position of the reference level:

Scale > Ref Position

DISP:WIND:TRAC:Y:RPOS — Sets the position of the reference line.



To set the number of trace scale divisions:

Scale > Divisions

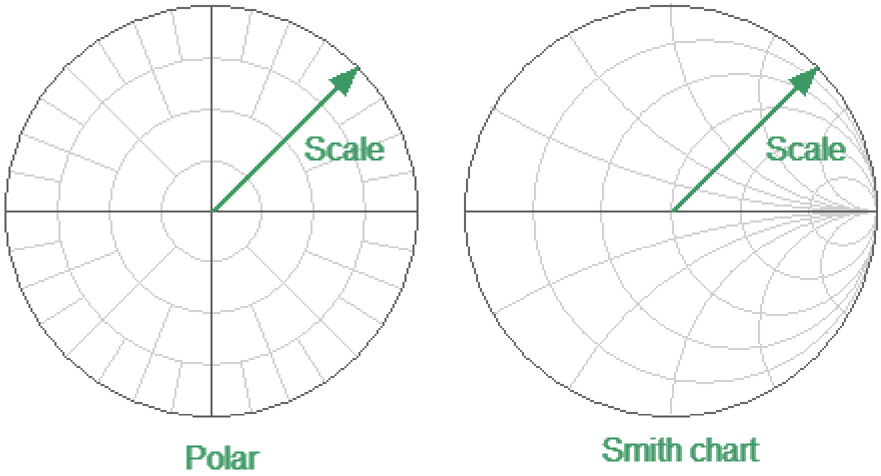
NOTE: The number of scale divisions affects all traces of the channel.

DISP:WIND:Y:DIV — Sets the number of the vertical scale divisions.

NOTE	The trace scale, value of the reference level, and reference level position can be set using the mouse (See Quick Setting Using a Mouse).

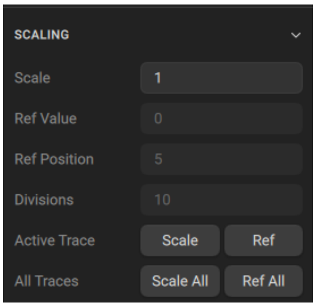
Circular Scale

For Polar formats the outer circle value can be set (See figure below).



Circular Scale

The scale of each trace can be set independently. The trace to which the function is applied must be preselected as active (See Selection of Active Trace/Channel).



To set the scale of the circular graphs, use the following softkeys:

Scale > Scale

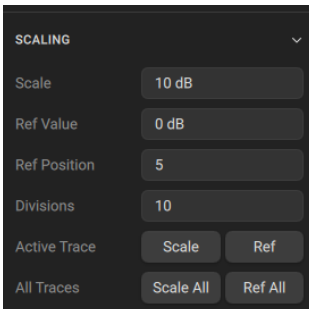
DISP:WIND:TRAC:Y:PDIV	Sets or reads out the trace scale. Sets the full scale value.

Automatic Scaling

The automatic scaling function automatically adjusts the trace scale so that the trace of the measured value fits into the diagram entirely.

In rectangular format, two parameters are adjustable: scale division and reference level position. In circular format, the outer circle value is adjusted.

The function can be applied to the active trace or to all traces of the active channel.



To automatically select the scale of the active trace:

Scale > Active Trace [Scale]

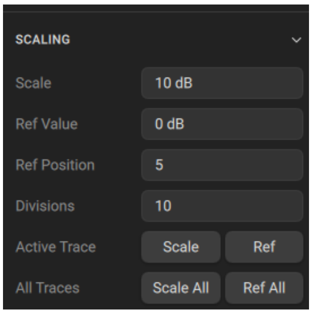
To automatically select the reference level of all traces of the active channel:

Scale > All Traces [Scale All]

DISP:WIND:TRAC:Y:AUTO	Executes the auto scale function for the trace.

Reference Level Automatic Selection

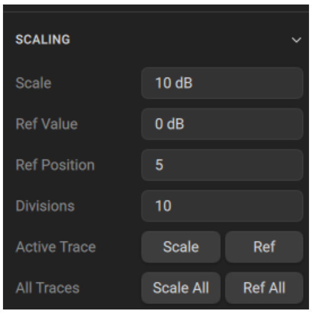
This function automatically selects the reference level in rectangular coordinates. After selection, the trace of the measured value shifts vertically so that the reference level crosses the trace in the middle. The scale division is unaffected. The function can be applied to the active trace or to all traces of the active channel.



To automatically select the reference level of the active trace:

Scale > Active Trace [Ref]

DISP:WIND:TRAC:Y:RLEV:AUTO	Executes the auto reference function for the trace.



To automatically select the reference level of all traces of the active channel:

Scale > All Traces [Ref All]

Electrical Delay Setting

The electrical delay function compensates for the electrical delay of the trace measurement. This function is useful during measurements of phase deviations from linear, for example.

If the electrical delay setting is other than zero, the S-parameter value will be corrected in accordance with the following formula:

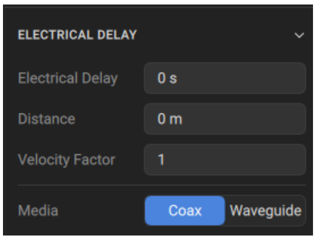
$$S = S_{meas} \cdot e^{j \cdot 2\pi \cdot f \cdot t},$$

where f — frequency, Hz,

t — electrical delay, sec.

The electrical delay can be specified in seconds or as an equivalent length (meters, feet, inches). When the equivalent length is used, it is also possible to select media (coax or waveguide) and to set the velocity factor and WG Cutoff (for waveguide only).

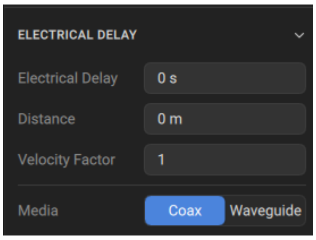
The electrical delay is set for each trace independently. The trace to which the function is applied must be preselected as active (See Selection of Active Trace/Channel).



To set the electrical delay in seconds:

Scale > Electrical Delay > Electrical Delay

CALC:CORR:EDEL:TIME	Sets or reads out the value of the electrical delay.



To set the electrical delay to an equivalent length:

Scale > Electrical Delay > Distance

CALC:CORR:EDEL:DIST — Sets or reads out the value of the equivalent distance in the electrical delay function.

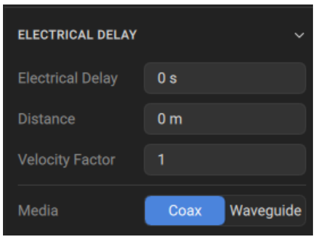
To set the units of equivalent length, use the following softkeys:

Scale > Electrical Delay > Distance Units

Select unit:

- **Meters**
- **Feet**
- **Inches**

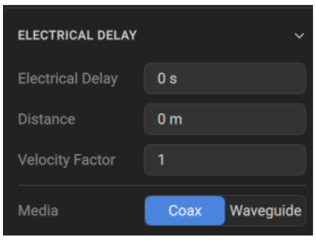
CALC:CORR:EDEL:DIST:UNIT — Sets or reads out the distance units in the electrical delay function.



To set the transmission line type:

Scale > Electrical Delay > Media [Coax | Waveguide]

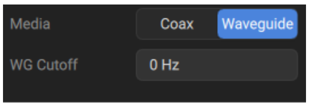
CALC:CORR:EDEL:MED — Sets or reads out the type of media in the electrical delay function.



To enter the velocity factor value:

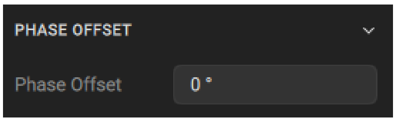
Scale > Electrical Delay > Velocity Factor

CALC:CORR:EDEL:RVEL — Sets or reads out the value of the velocity factor used to calculate between delay and distance in the electrical delay function.

	To enter the WG Cutoff value, use the following softkeys: **Scale > Electrical Delay > WG Cutoff**
CALC:CORR:EDEL:WAV:CUT	Sets or reads out the value of the waveguide cutoff frequency in the electrical delay function if the type of media set to the **WAVeguide** by the command CALC:CORR:EDEL:MED.

Phase Offset Setting

The phase offset function adds the constant offset to the phase of a trace. The value of the phase offset is set in degrees for each trace independently. The trace must be activated before setting the phase offset.



To set the phase offset:

Scale > Phase Offset

CALC:CORR:OFFS:PHAS	Sets or reads out the value of the phase offset.

Chart Zooming

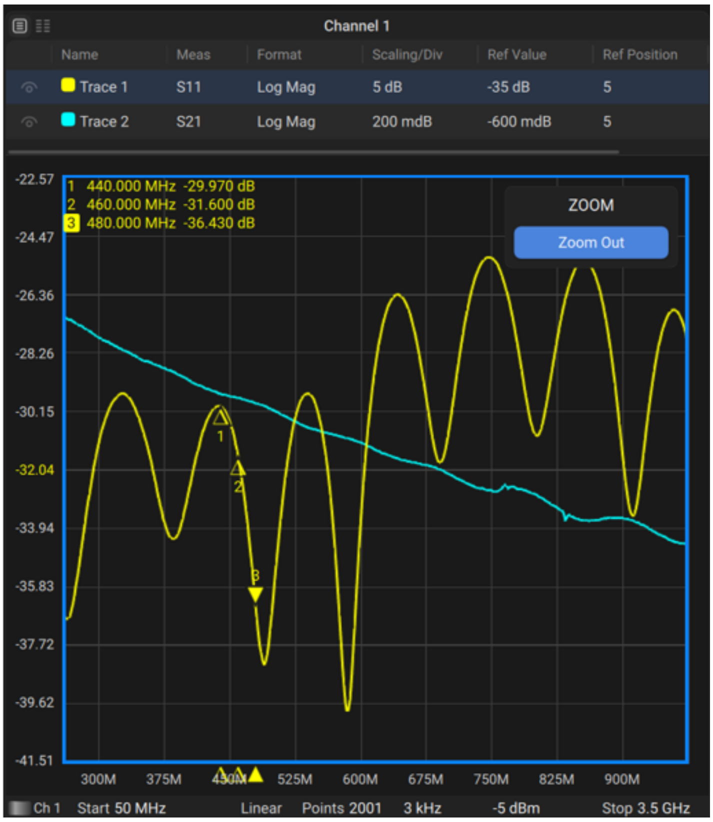


Chart Zooming

NOTE	The zooming mode does not affect the sweep parameters. For example, if the sweep range is set from 300 kHz to 9 GHz, the measurements will be performed withing the specified sweep range and will not be limited to the zoomed area.

Chart Zooming In/Out

1. Click and drag over the area to zoom in on.

2. If you want to zoom the chart out, click on the **Zoom Out** button in the right upper corner of the channel window.

   NOTE The chart will be displayed at its default scale.

Measurement Optimization

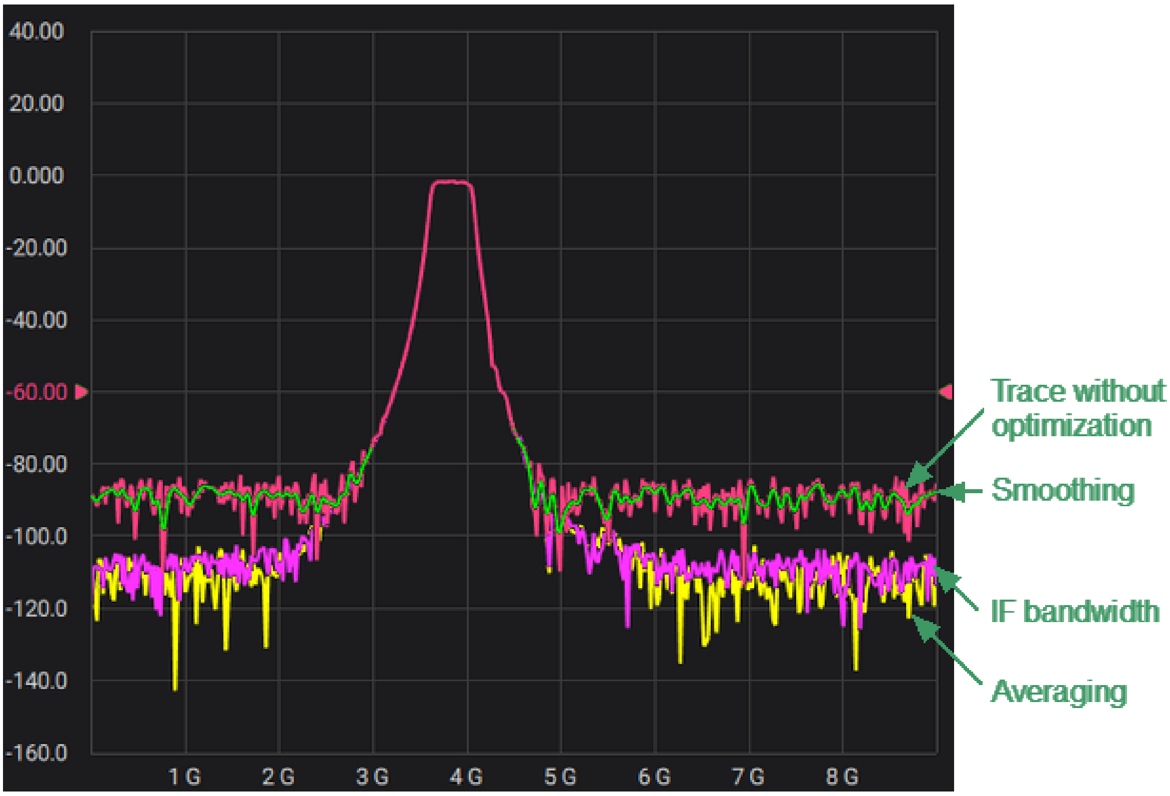
This section describes ways to optimize the measurement:

- Narrowing the IF bandwidth of measurement receivers increases the signal-to-noise ratio and extends the dynamic range of measurements. This increases the value of the sweep time. For a detailed description, see IF Bandwidth Setting.

- Averaging allows to increase the signal-to-noise ratio and extend the dynamic range of the measurements. Averaging does not increase the value of the sweep time, but the averaging result is complete after N sweeps, where N is an averaging factor. For a detailed description, see Averaging Setting.

- Smoothing does not change the dynamic range of the measurements but reduces the noise emissions of the signal. For a detailed description, see Smoothing Setting.

The figure below shows an example of applying different optimization methods to the signal: the IF bandwidth is reduced by a factor of 10, averaging factor is set to 100, and smoothing is applied with an aperture of 2%.



Example of the Application of Different Measurement Optimization

IF Bandwidth Setting

The IF bandwidth setting selects the bandwidth of the receivers. The IF bandwidth value can be selected from the following series: 1 Hz, 1.5 Hz, 2 Hz, 3 Hz, 5 Hz, 7 Hz, 10 Hz, 15 Hz, 20 Hz ... 200 kHz, 300 kHz.

Narrowing the IF bandwidth increases the signal-to-noise ratio and extends the dynamic range of measurements. Narrowing the IF bandwidth by 10 will nominally extend the dynamic range by 10 dB. Narrowing the IF bandwidth also increases the measurement time.

Setting IF bandwidth

The IF bandwidth is set for each channel independently.

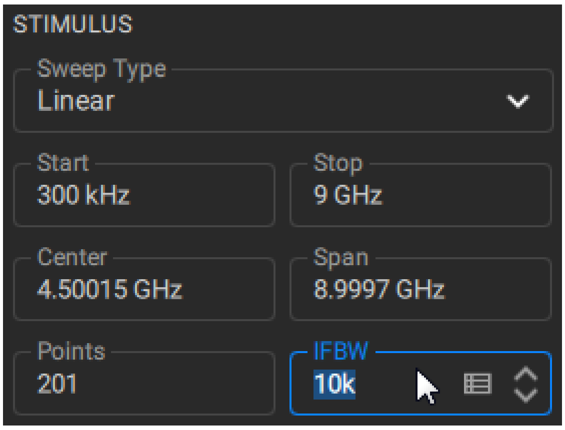
---

1. Select the channel.

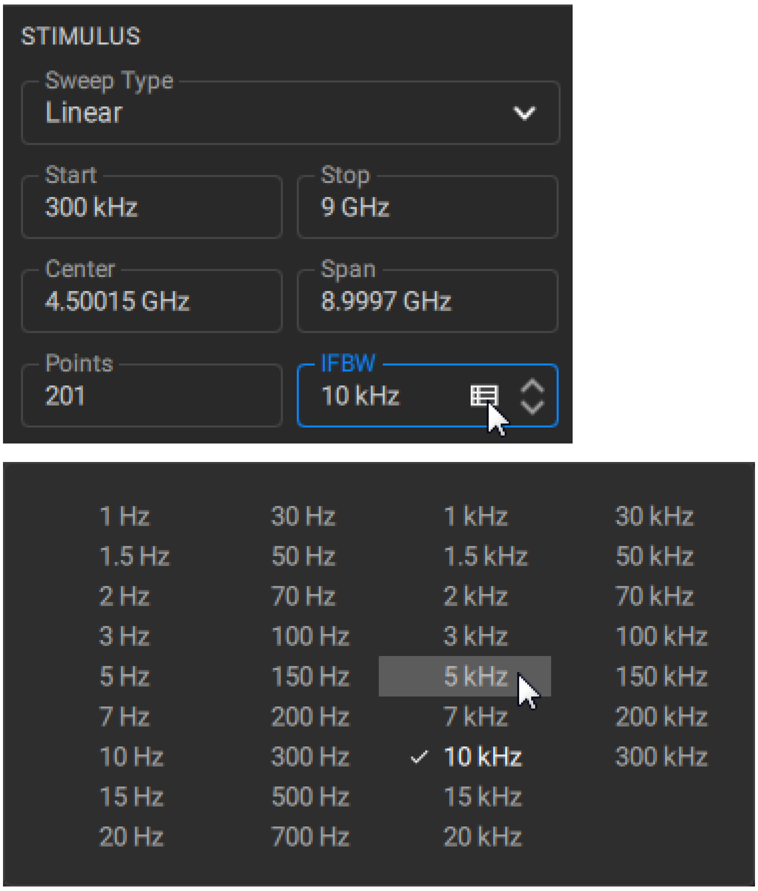
2. Click on the **Channel** button in the main menu.

3. Enter the numerical values of IF bandwidth:

   - Click on the **IFBW** textbox in the submenu and enter the numerical value using keyboard or spinbox (see figure below).



   - Click on the icon in **IFBW** textbox in the submenu and select the numerical value from the list in pop-up window (see figures below).

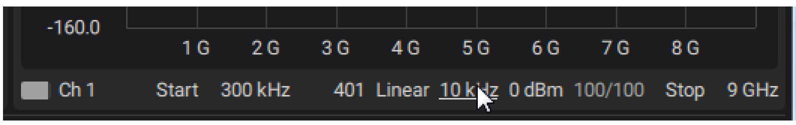


SCPI	SENSe:BANDwidth (SENSe:BWIDth)

NOTE

The IF bandwidth can be setting in the channel status bar (see figure above):

- Right click on the respective field in the channel status bar and enter the numerical values using keyboard or spinbox.
- Left click on the respective field in the channel status bar and select the numerical value from the list in pop-up window:



Averaging Setting

Averaging of each measurement point is performed over several sweeps. The benefits of the averaging function are similar to those of IF bandwidth narrowing. It increases the signal-to-noise ratio and extends the dynamic range of measurements.

Averaging of each measurement point is made across multiple sweeps in accordance with the following formula:

$$\begin{cases} M_i = S_i, \quad i = 0 \\ M_i = \left(1 - \frac{1}{n}\right) \cdot M_{i-1} + \frac{S_i}{n}, \quad i > 0, n = min\,(i + 1,\ N) \end{cases}$$

where M_i — i-th sweep averaging result,

S_i — i-th sweep measurement parameter (S-parameter) value,

N — averaging factor from 1 to 999; the higher the factor value, the stronger the averaging effect.

When the averaging function is enabled, the current number of iterations and the averaging factor, e.g. «9/10», will appear in the channel status bar. The averaging process is considered stable when the two numbers are equal.

Setting Averaging

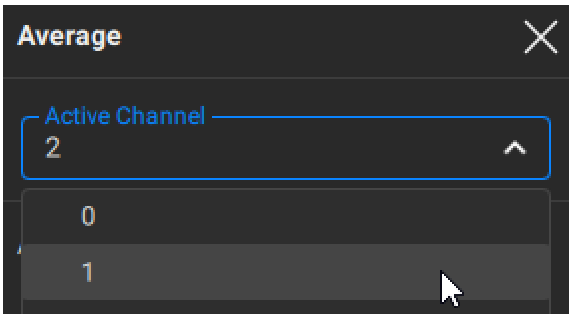
Averaging should be set for each channel individually.

---

1. Click the **Average** button in the main menu.

2. Select the channel in the **Active Channel** drop-down list in the submenu.

   NOTE You can quickly activate the channel by clicking on it in the channel window.



3. Toggle **AVERAGING** ON in the AVERAGING accordion in in the submenu.

4. Click the **Average Factor** textbox in the submenu and enter the number of sweeps for averaging.

NOTE After entering the **Average Factor**, the **Averaged** field will automatically display the current number of iterations in real time.

SCPI	SENSe:AVERage, SENSe:AVERage:COUNt

Smoothing Setting

Smoothing averages the adjacent points of the trace by the moving window. The window aperture is set as a percent of the total number of trace points.

Smoothing does not increase the dynamic range of the Analyzer, nor does it increase measurement time. Smoothing helps to reduce noise bursts.

Setting Smoothing

Smoothing is set for each trace independently.

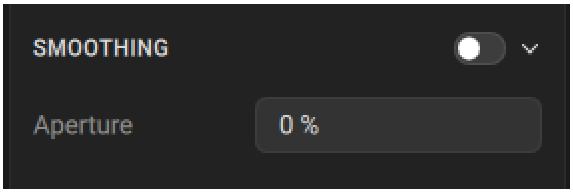
1. Select the channel in the **Active Channel** drop-down list in the submenu.

   NOTE You can quickly activate the channel by clicking on it in the channel window.

2. Select the trace in **Active Trace** drop-down list in the submenu.

   NOTE You can quickly activate the trace by clicking on it in the trace manager.

3. Toggle **TRACE SMOOTHING** ON in the TRACE SMOOTHING accordion in in the submenu.

4. Click the **Smoothing Aperture** textbox in the submenu and enter the numerical values.

SCPI	CALCulate:SMOothing, CALCulate:SMOothing:APERture

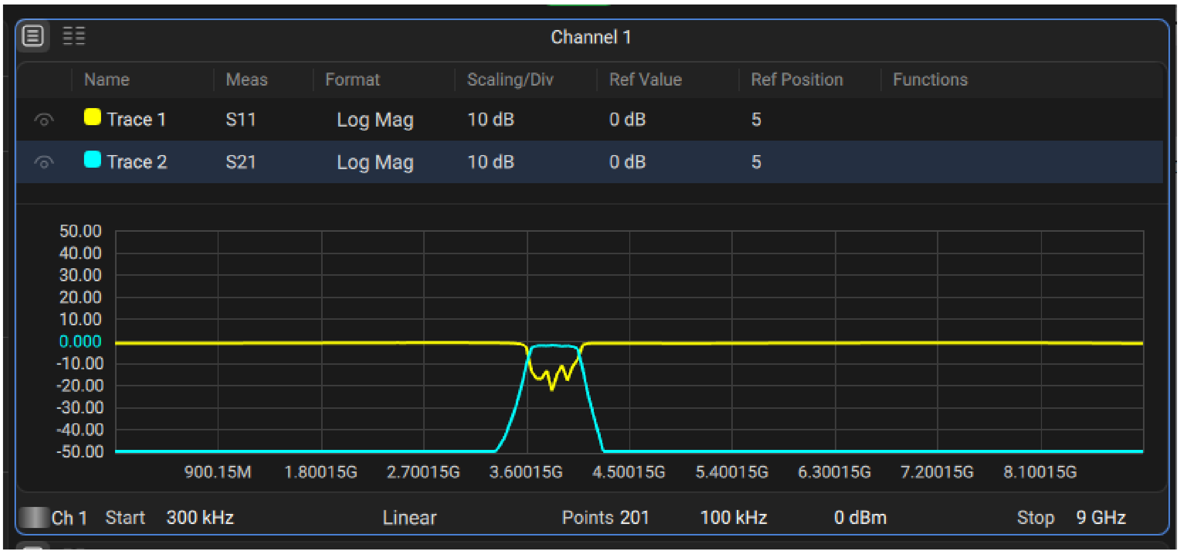
Quick Settings Using a Mouse

This section describes mouse operations, which allows to set the channel parameters quickly and easily. In a channel window, when hovering over the field where a channel parameter can be modified, the mouse pointer will change its icon to indicate edit mode. In text and numerical fields, edit mode will be indicated by underlined symbols.

NOTE	The mouse operations described in this section help to adjust the most frequently used settings.

Active Channel Selection

The active channel can be selected when two or more channel windows are open. The border line of the active window will be highlighted in a light color. To activate another window, click inside its area.

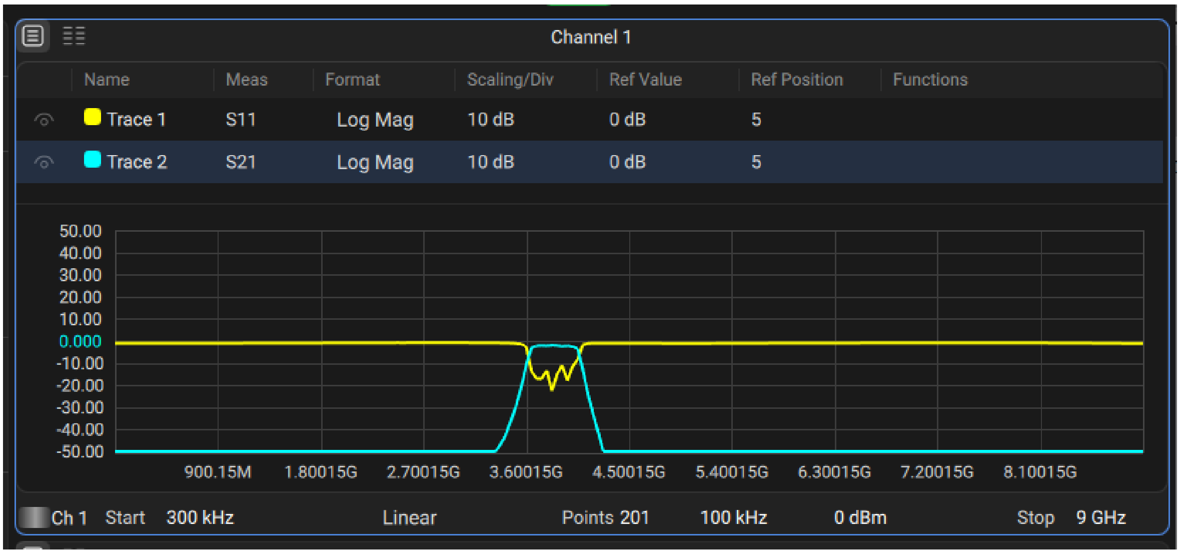


Active Trace/Channel Selection

The active channel can be selected using softkeys (See Selection of Active Trace/Channel).

Active Trace Selection

The active trace can be selected if the active channel window contains two or more traces. The active trace name is highlighted. To activate a trace, click on the required trace status line, or on any item (trace, marker) having the same color.

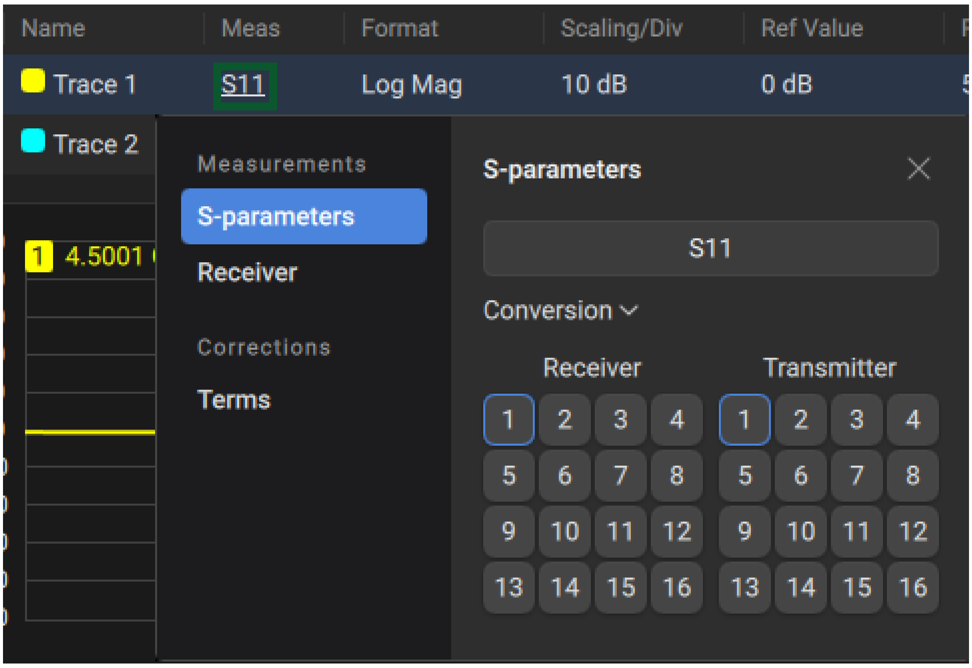


Active Trace Selection

Active trace can be selected using softkeys (See Selection of Active Trace/Channel).

Measured Parameter Setting

To assign the measured parameters (S11, S21) to an active trace, click on the S-parameter name in the trace status line and select the required parameter in the drop-down menu.



Measured Parameter Setting

Display Format Setting

To select the trace display format, click on the display format field in the trace status line and select the desired format in the drop-down menu.

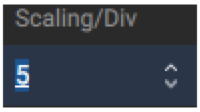
Display Format Setting

The display format can be set using softkeys (See Format Setting).

Trace Scale Setting

The trace scale, also known as the vertical scale division value, can be set by either of two methods.

The first method: click on the trace scale field in the trace status line and enter the required numerical value.



Trace scale setting in the trace status line

The second method: move the mouse pointer over the vertical scale until the pointer icon becomes as shown in the figure. The pointer should be placed in the top or bottom parts of the scale, at approximately 10% of the scale height from the top or bottom of the scale. Left click and drag away from the scale center to enlarge the scale, or toward the scale center to reduce the scale.

Trace scale setting on the vertical scale

The trace scale can be set using softkeys (See Rectangular Scale).

Reference Level Setting

The value of the reference level can be set by either of two methods.

The first method: click on the reference level field in the trace status line and enter the required numerical value.

Reference level setting in the trace status line

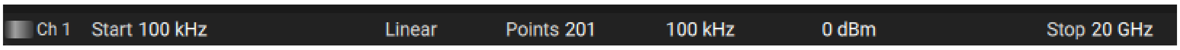
The second method: move the mouse pointer over the vertical scale until the pointer icon becomes as shown in the figure. The pointer should be placed in the center part of the scale. Left click and drag up to increase the reference level value, or down to reduce the value.

Reference level setting on the vertical scale

The value of the reference level can be set using softkeys (See Rectangular Scale).

Switching Between Start/Center and Stop/Span Modes

To switch between the modes, Start/Center and Stop/Span, click on the respective field of the channel status bar. Clicking the label «Start» changes it to «Center», and the label «Stop» will change to «Span».

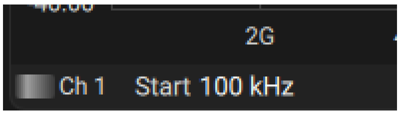


Channel Status Bar

The layout of the stimulus scale will be changed correspondingly. Switching between can be done in the Channel menu (See Sweep Range).

Start/Center Value Setting

Click on the start value at the bottom of the screen and either type the desired start or center value or using the arrows to change the value.

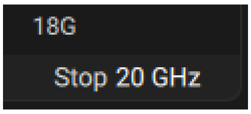


Setting the Start/Center value in the channel status bar

The Start/Center values can be set in the Channel menu (See Sweep Range).

Stop/Span Value Setting

Click on the stop value at the bottom right of the screen and either type the desired stop or span value or using the arrows to change the value.

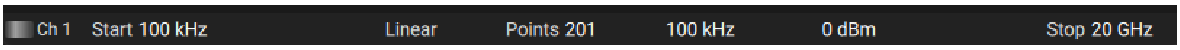


Setting the Stop/Span value in the channel status bar

The Stop/Span values can be set in the Channel menu (See Sweep Range).

Number of Points Setting

To enter the number of points, activate the respective field in the channel status bar by clicking the numerical value.

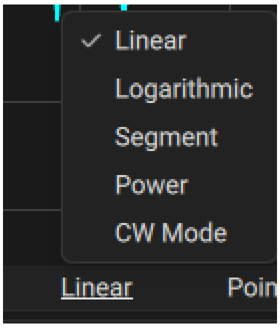


Setting the number of points value in the channel status bar

The number of points can be set in the Channel menu (See Number of Points).

Sweep Type Setting

To set the sweep type, left click on the respective field in the channel status bar and select the required type in the drop-down menu.

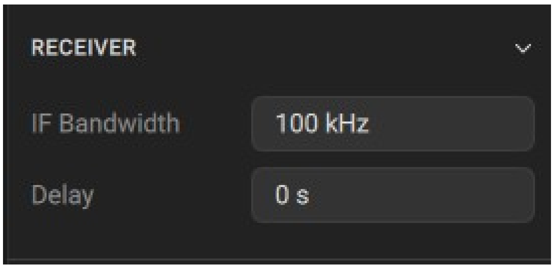


Setting the sweet type value in the channel status bar

The sweep type can be selected in the Channel menu (See Sweep Type).

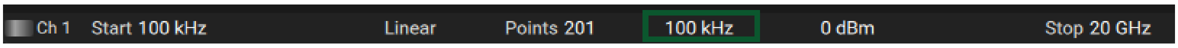
IF Bandwidth Setting

IF bandwidth can be set by by entering the value using numerical keys of the keyboard in the channel status bar or in the Channel menu.



IF Bandwidth Setting in drop-down menu

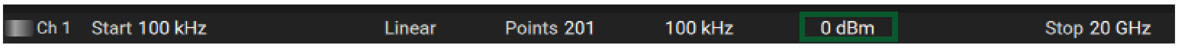
To enter the IF bandwidth, activate the respective field in the channel status bar by left clicking.



Setting the IF Bandwidth in the channel status bar

Power Level/CW Frequency Setting

To enter the Power Level/CW Frequency, activate the respective field in the channel status bar by clicking the numerical value. The parameter displayed in the field depends on the current sweep type: in frequency sweep mode, the power level value can be entered; in power sweep mode, the CW frequency value can be entered.



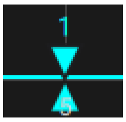
Setting the Power Level/CW Frequency in the channel status bar

The Power Level and CW Frequency can be set using softkeys (See Stimulus Power and CW Frequency).

Marker Stimulus Value Setting

The marker stimulus value can be set by either a click and drag operation, or by entering the value using numerical keys of the keyboard.

To drag the marker, first move the mouse pointer on one of the marker indicators until it becomes as shown in the figures below:



Setting the marker value using drag and drop

To enter the numerical value of the stimulus, activate its field by clicking it in the marker data line.

Setting the marker value in the marker data line

The marker stimulus value can be set in the Marker menu (See Marker Stimulus Value Setting).

Calibration and Calibration Kits

Measurement accuracy is affected by errors introduced by the Analyzer and measurement setup. The nature of these errors is varied — some are systematically repeated, and some are random. Calibration is a process used to evaluate systematically repeated errors and mathematically exclude them from the measurement results in the correction process.

NOTE	Be sure to properly calibrate if accurate measurements are required. Only a properly calibrated instrument provides the accuracy specified in the data sheet.

The section describes information about calibration, calibration kits and automatic calibration module (ACM):

- General information about calibration (See General Information).
- Working with calibration standards and calibration kits (See Calibration Standards and Calibration Kits).
- Calibration method and procedures (See Calibration Methods and Procedures).
-
-
-
- Working with the automatic calibration module (ACM), which allows for simplification and speeding up of the analyzer calibration process.

General Information

This section details general information about calibration:

- Guidelines for calibration (See Basic Calibration Guidelines).
- Description of measurement errors (See Measurement Errors).
- Error models (See Error Model).
- Calibration steps (See Calibration Steps).

Basic Calibration Guidelines

Follow the guidelines below to perform calibration correctly and reduce accidental errors. Observance of the guidelines will ensure the specified accuracy of the device.

General Guidelines

- Select all fixtures for connecting the DUT and assemble the measuring setup before starting the calibration. Perform calibration in the plane passing through the connectors to which the DUT is connected.
- Calibrate the measuring setup at the same stimulus parameters (frequency range, number of points, stimulus power) at which measurements will be performed. Changing these parameters after calibration may significantly reduce the accuracy of the measurements.
- During calibration, do not set the IF bandwidth wider than planned for measurements.
- Choose a calibration kit according to the type and gender of the DUT connectors.
- The frequency range of the selected calibration kit must correspond to the range in which the calibration is performed.
- When choosing a calibration kit, note that for SOLT calibrations the most accuracy will be provided by the calibration kit, in which the parameters of the standards are most accurately defined. For TRL calibrations, the accuracy of the calibration is mainly determined by the quality of the standards manufacturing.
- The calibration kit selected in the Analyzer software must strictly correspond to the one actually used. The mismatch is unacceptable.
- For easy measurements, it is possible to create custom calibration kits from the available standards or specially manufactured calibration kits to solve specific measuring tasks. To include a standard in a calibration kit, calculate or measure its parameters using a high precision measuring tool. Create a description in the form of a model of standard or S-parameter table of standard and download this description to the analyzer software.
- The choice of calibration method depends on the measurement being performed, its accuracy requirements, the permissible calibration labor intensity, and the availability of calibration kits.
- For the SOLT calibrations, it is recommended to use an ACM (Automatic Calibration Module) to reduce:
  - the labor intensity of the calibration without loss of accuracy
  - wear of connectors

    - operator errors

- If an additional component (cable, attenuator, adapter) is added to the measurement setup after calibration, recalibrate. Instead of recalibration, it is possible to use the de-embedding function or the port extension function to compensate for the added electrical length (delay) and losses.

Recommendations for Reducing Random Measurement Errors

- To reduce errors introduced by the instrument noise of the Analyzer, it is recommended to increase the source power of the stimulus signal, narrow the IF bandwidth, and apply averaging over several measurement sweep values.

- To reduce errors in the temperature drift of the electrical characteristics of the Analyzer and the components of the measuring setup, it is recommended:
    - To perform measurements in a room with a stable, controlled temperature, at which the technical characteristics of the analyzer are guaranteed.
    - To recalibrate if the room temperature has changed significantly after calibration.
    - To warm-up the analyzer for a time determined in the specification before starting the calibration.
    - To keep the calibration standards unpacked in the room where the measurements are taken to stabilize the parameters, before starting the calibration.

- To reduce the connector repeatability errors, it is recommended:
    - To apply proper connector care — connectors must be good and clean (See Connector Care).
    - To use a special wrench with a standardized tightening torque, when connecting the DUT and calibration standards to measurement connectors.
    - To not change the position of the components of the measuring setup in space during or after calibration.
    - To recalibrate if setup components have been rearranged.

Measurement Errors

S-parameter measurements are influenced by various measurement errors, which can be broken down into two categories:

- systematic errors
- random errors

Random errors comprise errors such as noise fluctuations and thermal drift in electronic components, changes in the mechanical dimensions of cables and connectors subject to temperature drift, repeatability of connections, and cable bends. Random errors are unpredictable and hence cannot be estimated and eliminated in calibration. Random errors can be reduced by having the stimulus power at the correct setting, IF bandwidth narrowing, sweep averaging, maintaining a constant environment temperature, observance of the Analyzer warm-up time, careful connector handling, and avoiding cable bending after calibration.

Random errors and related methods of correction are not mentioned further in this section.

Systematic errors are errors caused by imperfections in the components of the measurement system (See Systematic Errors). Such errors occur repeatedly, and their characteristics do not change with time. Systematic errors can be determined and then reduced by performing a mathematical correction of the measurement results.

Calibration is the process of measuring precision devices with predefined parameters to determine systematic errors, and such precision devices are called **calibration standards**. The most used calibration standards are SHORT, OPEN, and LOAD.

The process of mathematical compensation of the systematic errors is called **error correction**.

Systematic Errors

The systematic measurement errors of the Analyzer are divided into the following categories according to their source:

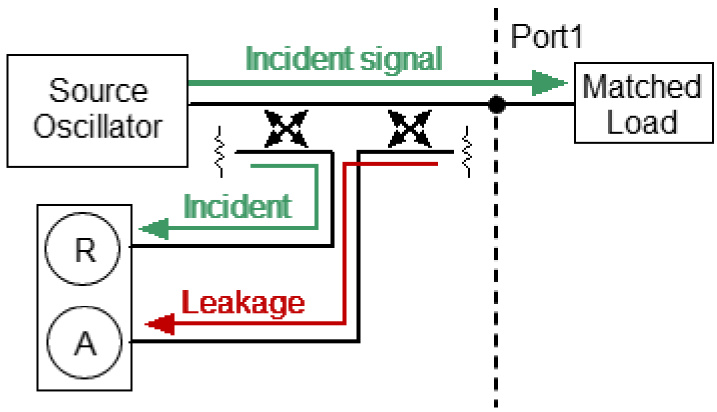
- directivity
- source match
- load match
- reflection tracking
- transmission tracking
- isolation

The measurement results before error correction are called **uncorrected**.

The residual values of the systematic measurement errors after error correction are called **effective**.

Directivity Error

A directivity error (**Ed**) is caused by incomplete separation of the incident signal from the reflected signal by the directional coupler in the source port. In this case, part of the incident signal energy enters the receiver of the reflected signal. Directivity errors do not depend on the characteristics of the DUT, and usually have a greater effect on reflection measurements.

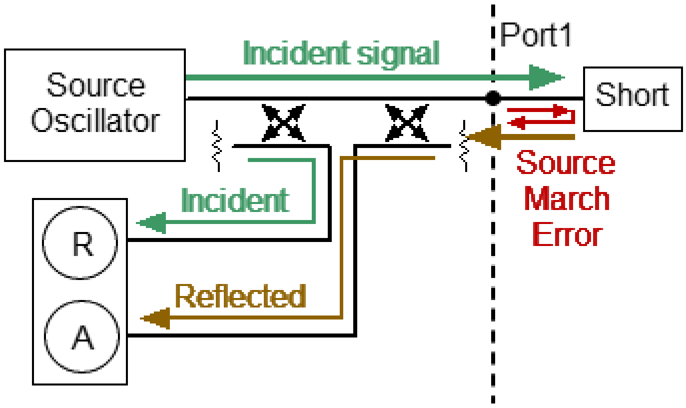


Directivity error

Source Match Error

A source match error (**Es**) is caused by a mismatch between the source port and the input of the DUT. In this case, part of the signal reflected by the DUT reflects at the source port and re-enters the input of the DUT. The error affects both reflection measurement and transmission measurement. Source match errors depend on the difference between the input impedance of the DUT and test port impedance when it functions as a signal source.

Source match errors heavily affect measurements of a DUT with poor input matching.

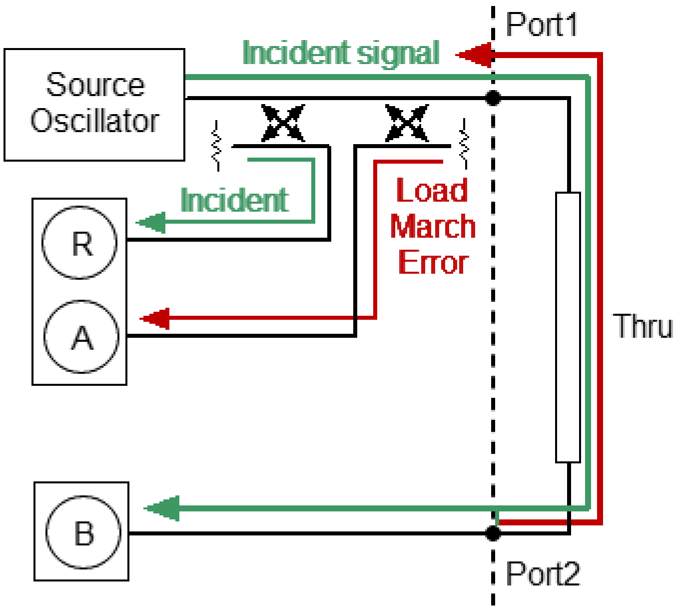


Source match error

Load Match Error

A load match error (**El**) is caused by a mismatch between the receiver port and the output of the DUT. In this case, part of the signal transmitted through the DUT reflects at the receiver port and returns to the output of the DUT. The error occurs during transmission measurements and reflection measurements (for a two-port DUT). Load match errors depend on the difference between output impedance of the DUT and test port impedance when used as a signal receiver.

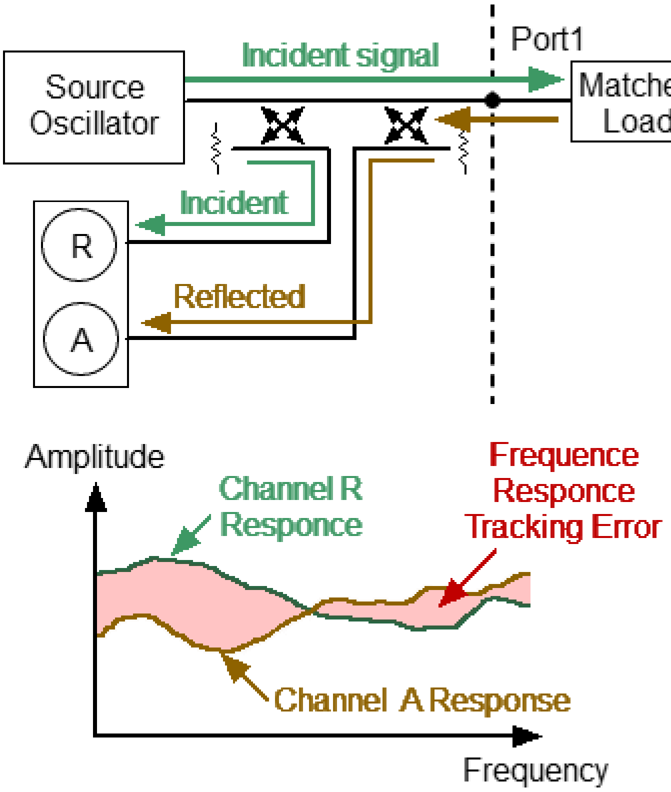
In transmission measurements, the load match error has considerable influence if the output of the DUT is poorly matched. In reflection measurements, the load match error has considerable influence in cases of poor output match and low attenuation between the output and input of the DUT.



Load match error

Reflection Tracking Error

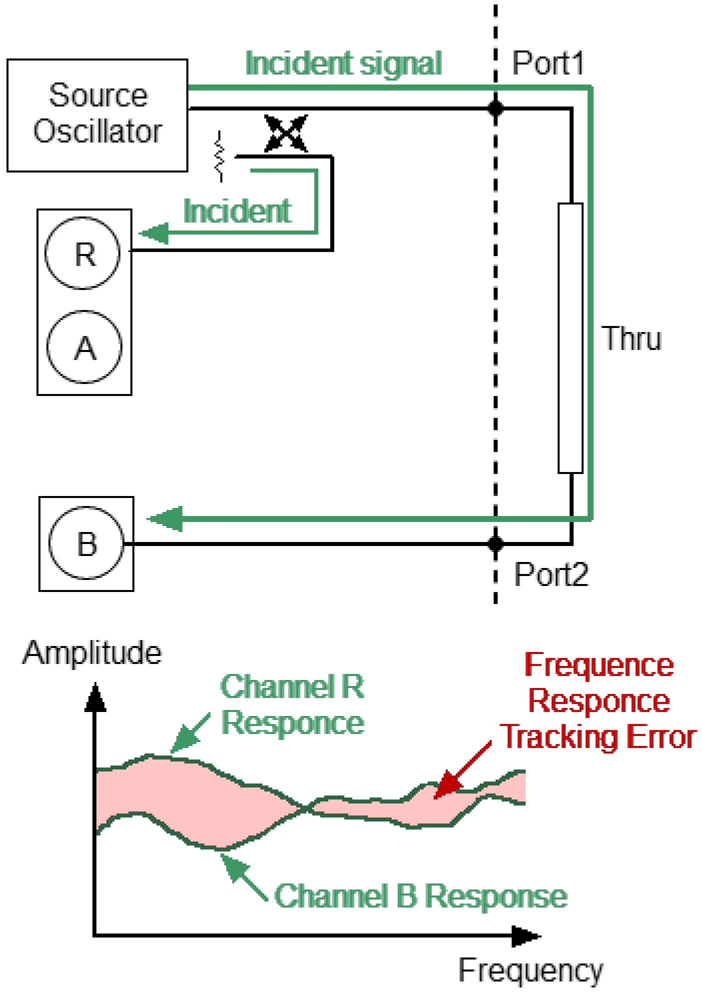
A reflection tracking error (**Er**) is caused by differences in frequency response between the test receiver and the reference receiver of the source port during reflection measurement.



Reflection tracking error

Transmission Tracking Error

A transmission tracking error (**Et**) is caused by differences in frequency response between the test receiver of the receiver port and the reference receiver of the source port during transmission measurement.

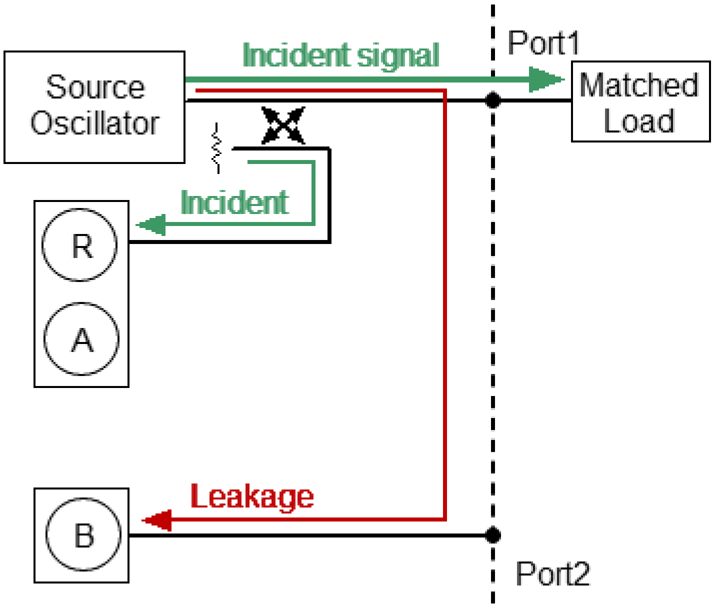


Transmission tracking error

Isolation Error

Isolation error (**Ex**) is caused by a leakage of the signal from the source port to the receiver port bypassing the DUT.

The Analyzer has very good isolation, which allows us to ignore this error for most measurements. Isolation error measurement is an optional step in all types of calibration.



Isolation error

Error Model

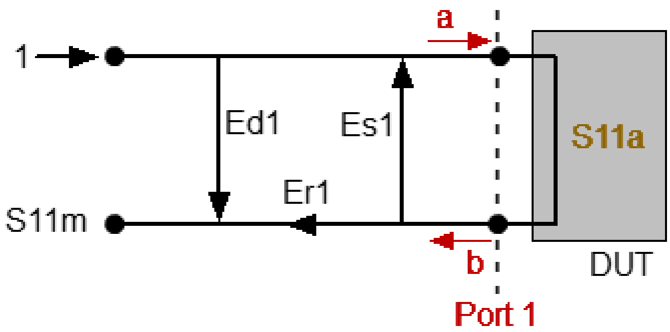
The error model in the form of signal (directed) graphs is used to analyze systematic errors of the Analyzer.

This section describes following error models:

- One-Port Error Model
- One-Path Two-Port Error Model

One-Port Error Model

Only one port of the Analyzer is used when performing reflection measurements. The signal flow graph of errors for Port 1 is represented in the figure below. For Port 2, the signal flow graph of the errors will be similar.



a — incident wave, b — reflected wave

S11a — reflection coefficient actual value

S11m — reflection coefficient measured value

One-port error model

The measurement result at Port 1 is affected by the following three systematic error terms:

- **Ed1** is directivity.
- **Es1** is source match.
- **Er1** is reflection tracking.

For normalization, the stimulus value is taken equal to 1. All the values used in the model are complex.

After determining all the three error terms — **Ed1, Es1, Er1** — for each measurement frequency by means of a **full one-port calibration**, it is possible to calculate (mathematically eliminate the errors from the measured value S11m) the actual value of the reflection coefficient S11a.

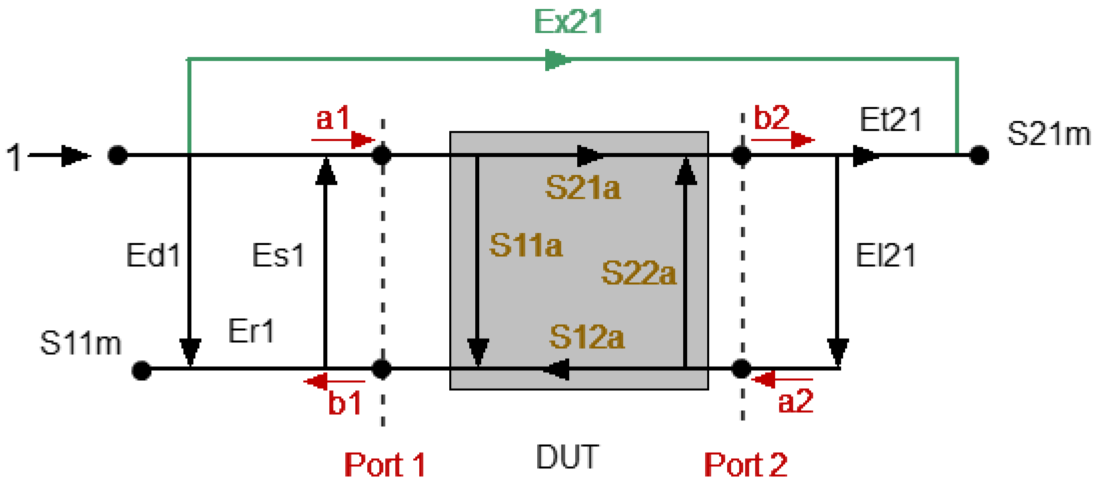
There are simplified methods, which eliminate the effects of only one or two of the three systematic errors.

For a detailed description of calibration methods, see Calibration Methods and Procedures.

One-Path Two-Port Error Model

For a one-path measurement of the reflection coefficient and the transmission coefficient of a two-port DUT, the two ports of the Analyzer are used.

The signal flow graph of errors effect in a one-path two-port system is represented in the figure below.



a1, a2 — incident waves, b1, b2 — reflected waves

S11a, S21a — actual value of DUT parameters

S11m, S21m — measured DUT parameters values

One-path two-port error model

For normalization the stimulus value is taken equal to 1. All the values used in the model are complex. The measurement result in a one-path two-port system is affected by six systematic error terms.

These terms are also described in the table below.

Description	Error
Directivity	**Ed1**
Source match	**Es1**
Reflection tracking	**Er1**
Transmission tracking	**Et1**
Load match	**El1**
Isolation	**Ex1**

After determination of the four error terms (Ed1, Es1, Er1, Et1) for each measurement frequency by means of a one-path two-port calibration, it is possible to calculate the true value of the S11a. The calibration does not take into account El1 error and takes into account optional Ex1, that is why the measured value of S21m will become closer to the true value of S21a with the improvement of the source match and increasing the isolation.

For a detailed description of calibration methods, see Calibration Methods and Procedures.

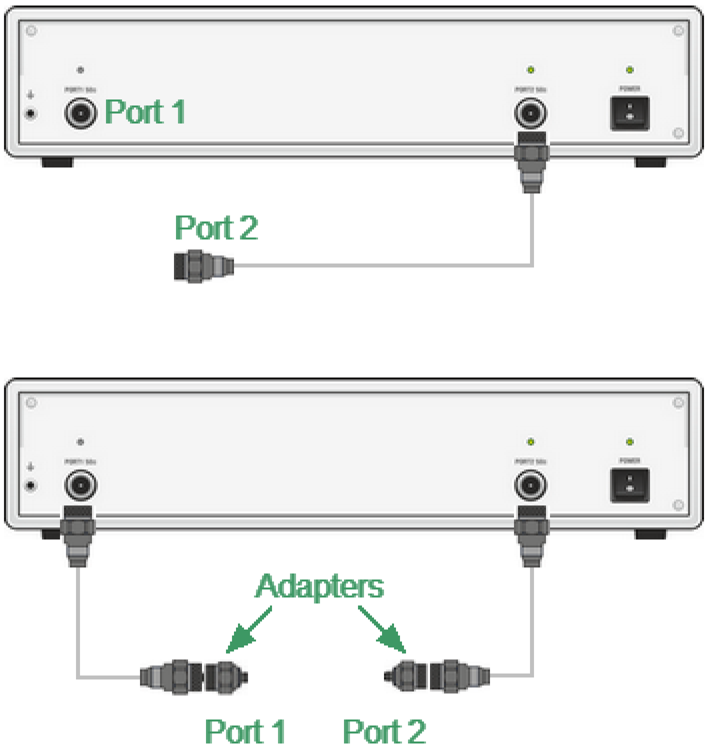
Analyzer Test Port Definition

The test ports of the Analyzer are defined by means of calibration. The test port is a connector accepting a calibration standard in the process of calibration.

A type-N, 3.5 mm NMD connector on the front panel of the Analyzer will be the test port if calibration standards are connected directly to it.

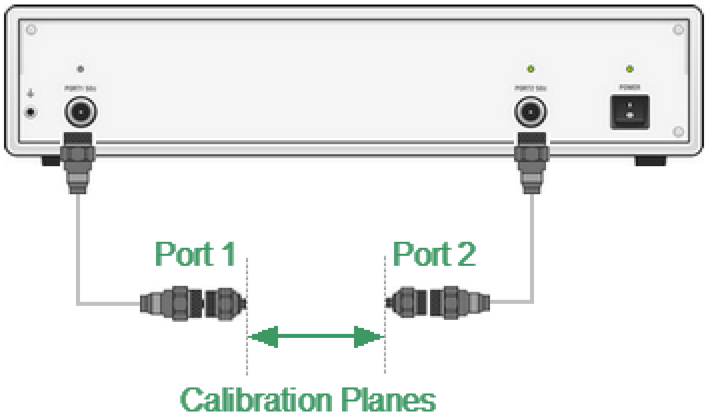
Sometimes it is necessary to connect coaxial cables and/or adapters to the connector(s) on the front panel to interface with a DUT of a different connector type. In such cases, calibration standards are connected to the connector of the cable or adapter.

The figure below represents two cases of test port definition for measurements. The use of cables and/or adapters does not affect the measurement results if they are integrated into the process of calibration.



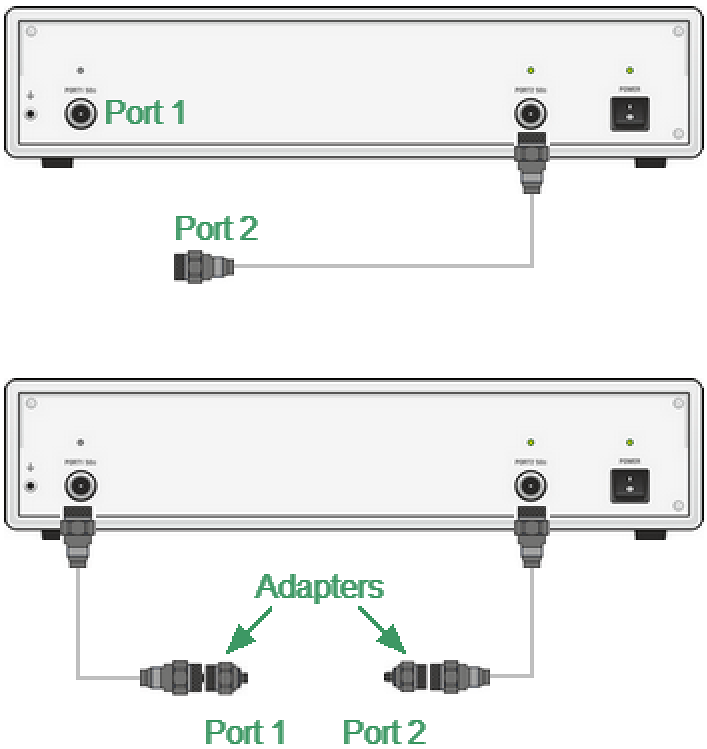
Test port defining

The term calibration plane is used in some cases. The calibration plane is an imaginary plane located at the ends of the connectors, which accept calibration standards during calibration.



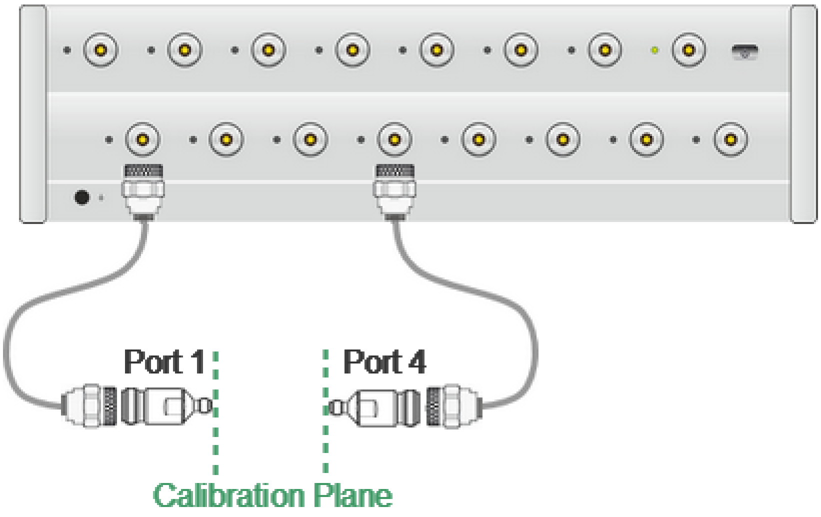
Calibration Planes

The figure below represents two cases of test port definition for measurements. The use of cables and/or adapters does not affect the measurement results if they are integrated into the process of calibration.



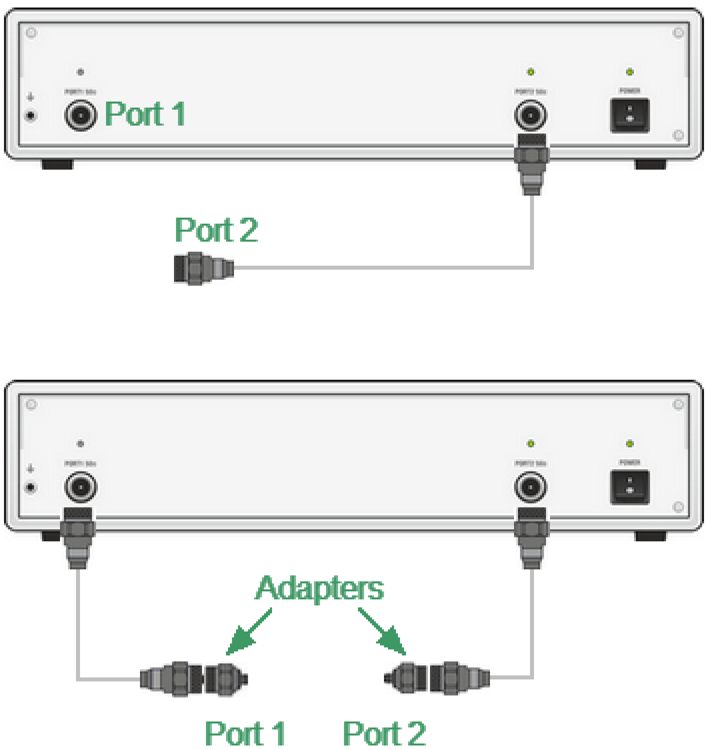
Test port defining

The term calibration plane is used in some cases. The calibration plane is an imaginary plane located at the ends of the connectors, which accept calibration standards during calibration.



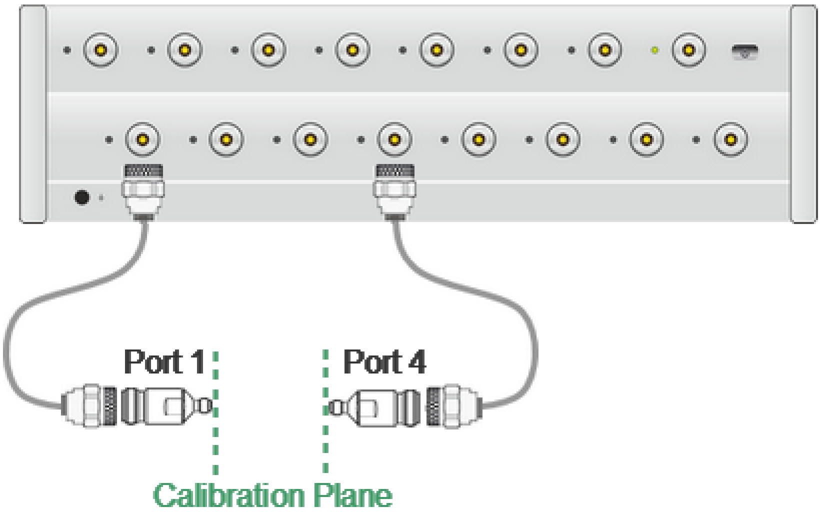
Calibration plane

The figure below represents two cases of test port definition for measurements. The use of cables and/or adapters does not affect the measurement results if they are integrated into the process of calibration.



Test port defining

The term calibration plane is used in some cases. The calibration plane is an imaginary plane located at the ends of the connectors, which accept calibration standards during calibration.



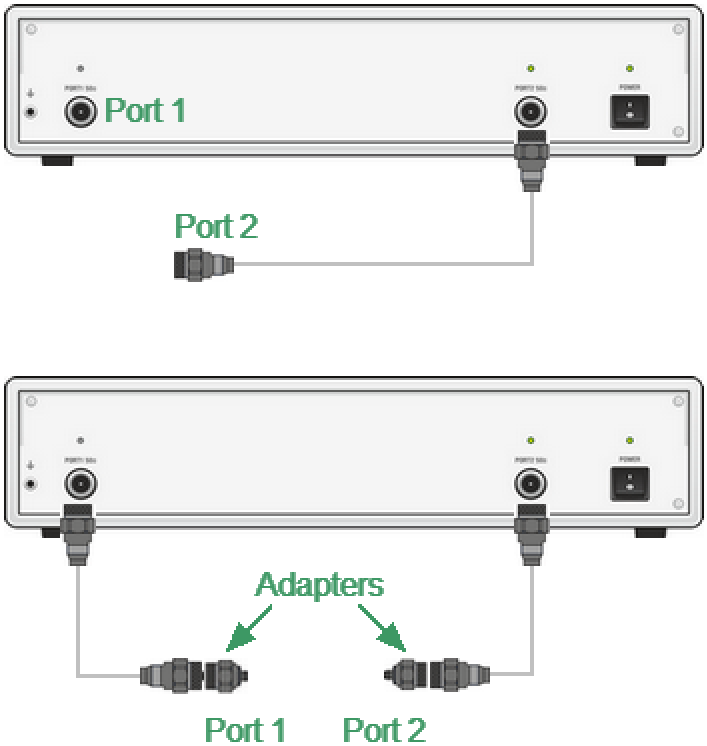
Calibration plane

The test ports of the Analyzer are defined by means of calibration. The test port is a connector accepting a calibration standard in the process of calibration.

A type-N, 3.5 mm NMD, 2.4 mm NMD or 1.85 mm NMD connector on the front panel of the Analyzer will be the test port if calibration standards are connected directly to it.

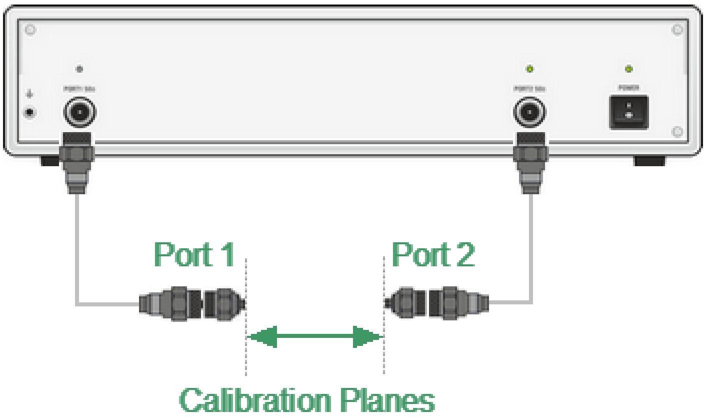
Sometimes it is necessary to connect coaxial cables and/or adapters to the connector(s) on the front panel to interface with a DUT of a different connector type. In such cases, calibration standards are connected to the connector of the cable or adapter.

The figure below represents two cases of test port definition for 4-port measurements. The use of cables and/or adapters does not affect the measurement results if they are integrated into the process of calibration.



Test port defining

The term calibration plane is used in some cases. The calibration plane is an imaginary plane located at the ends of the connectors, which accept calibration standards during calibration.



Calibration planes

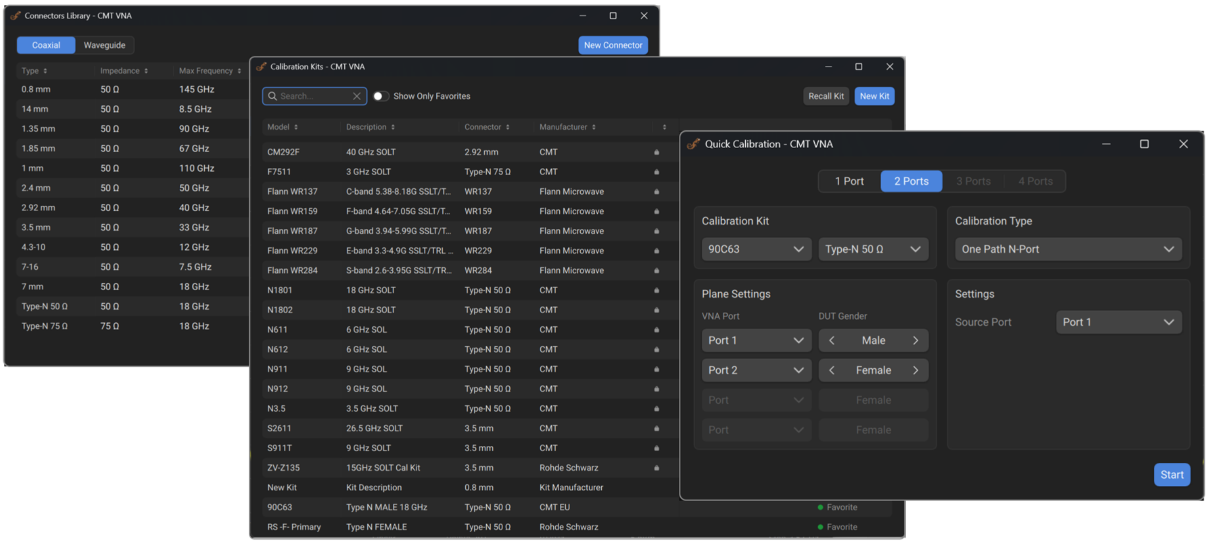
Calibration Steps

The process of calibration comprises the following steps:

- Selection of a calibration kit matching the connector type of the test port (See Calibration Standards and Calibration Kits). The calibration kit includes such standards as SHORT, OPEN, and LOAD with matched impedance. Magnitude and phase responses i.e. S-parameters of the standards are well known. The characteristics of the standards are represented in the form of an equivalent circuit model, as described in Circuit Model Standards.
- Selection of a calibration method (See Calibration Methods and Procedures) is based on the required accuracy of measurements. The calibration method determines which error terms of the model (or all of them) will be compensated.
- Measurement of the standards within a specified frequency range. The number of measurements depends on the type of calibration.
- The Analyzer compares the measured parameters of the standards against their predefined values. The difference is used for calculation of the calibration coefficients (systematic errors).
- The table of calibration coefficients is saved into the memory of the Analyzer and used for error correction of the measured results of any DUT.

Calibration is always made for a specific channel, as it depends on the channel stimulus settings — particularly on the frequency span. This means that a table of calibration coefficients is being stored for each individual channel.

Quick Calibration



Connectors Library, Calibration Kit Editor, and Quick Calibration Windows

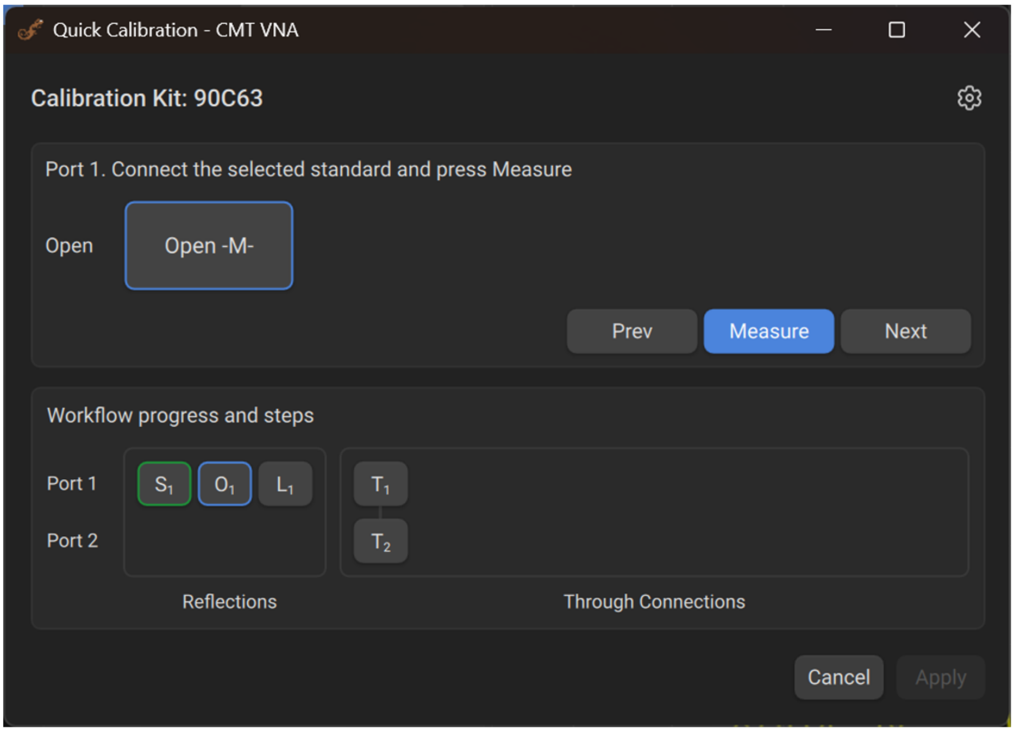
Select a calibration kit from the pull-down menu. If your kit does not appear in the pull-down, you will need to open the “Edit Kits…” menu and either favorite an existing kit or add a new one. To favorite an existing kit, move the mouse to the right of the kit description, until the ellipsis appears. Click on “Favorite” to allow this kit to be used. This selection process simplifies the calibration kit selection drop-down menu, populating it first with favorited kits and then all other kits.

Choose the type of gender of the DUT connectors for each port, and the type of calibration to be performed and push “Start”. The “Reduce reflection” and “Reduce transmission” choices reduce the number of calibration steps required at a very small cost to calibration accuracy.

“Reduce reflection” requires the user to choose a common port from the pull-down menu. When Full N-Port calibration is performed, the Open, Short, and Load standards will only be applied to the common port. The properties of the other ports are calculated from the 1-port calibration of the common port and the thru measurements. If “Reduce reflection” is not selected, the Open, Short and Load standards must be measured at all ports.

“Reduce transmission” also requires the selection of a common port. When the thru calibration is performed, the calibration routine will ask that the common port be connected to each of the other ports, one at a time. The remaining thru responses are calculated. This eliminates the need to make every possible pairing. Every possible pairing is N*(N-1)/2 or 6 pairings vs 4 for a 4-port calibration with reduced transmission mode. For 16 port calibration there are 120 pairings rather than 16.

Attach the calibration standards one at a time to each port (or only one port if “Reduce reflection” has been chosen) and press the “Measure” button for each. The calibration standards indicated on the screen will turn green after each measurement is complete. After all standards are measured, press the “Apply” button.



View of Quick Calibration using Cal Kit 90C63

All calibration is performed using the Calibration wizard. It offers a step-by-step procedure for the selected calibration type.

Calibration Standards and Calibration Kits

Calibration standard

Calibration standards are precision physical devices that serve as a calibration standard for the Analyzer.

Calibration standards have their own specific type (see Types of Calibration Standards), gender (see Gender of Calibration Standards), impedance, and definition.

The description of connector parameters is provided in Connectors Library. The detailed information about standards included in the calibration kit are provided in Standards Management in Calibration Kit.

Calibration standard definition is a mathematical description of its parameters (see Standards Definition). During calibration, the Analyzer measures standards and mathematically compares the results to the definitions of those standards. The comparison results are used to determine errors in the measurement system.

Calibration standard class is an application of the standard in a specific calibration method. For a detailed description of calibration standard classes, see Classes of Calibration Standards. A calibration standard belongs to a single class. Detailed instructions for assigning classes to standards in a calibration kit are provided in Classes Management.

Calibration standards can be combined into a calibration kit.

Calibration Kit

A calibration kit is a set of calibration standards with a specific connector type and specific impedance.

The Analyzer provides definitions of calibration kits produced by different manufacturers. The definitions of the calibration kits can be added, and the predefined kits can be modified.

The calibration kit editing procedure is described in Calibration Kits Management.

Types of Calibration Standards

Calibration standard type is a category of physical devices used to define the parameters of the standard. The Analyzer supports the following types of the calibration standards:

- OPEN
- SHORT
- LOAD
- THRU
- LINE

NOTE	The type of calibration standard should not be confused with its class. Calibration standard type is a part of the standard definition used for the calculation of its parameters. For example, a full single-port calibration (SOL) requires measuring three classes of standards: SHORT, OPEN, and LOAD. For the simplest SHORT class calibration, use a standard of the same type. The same goes for calibration of the OPEN and LOAD. However, a full, single-port calibration (SOL) can be performed with three SHORT calibration standards, for example. Thus, the class is related to the type of calibration, and correctly assigning calibration standards from the calibration set to classes ensures that the required calibration is performed step-by-step.

Gender of Calibration Standard

The gender of a calibration standard is typically denoted on the calibration standard label.

A calibration standard can be labeled either with:

- The gender of a calibration standard itself – –M– for male, –F– for female, and –G– for genderless type of standard.
- The gender of the analyzer port which the calibration standard is mated to – (m) for male and (f) for female port types.

For example, the same standard can be labeled as Short –F– or Short (m).

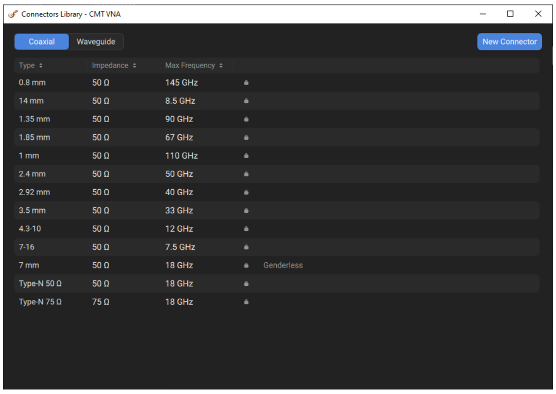
The gender of calibration standards is accounted for by the software. The Analyzer software uses the first type of designation in the Calibration Kit Manager: the gender of a calibration standard itself, denoted as **–M–** for male, **–F–** for female, and **–G–** for genderless type of standards.

Connectors Library

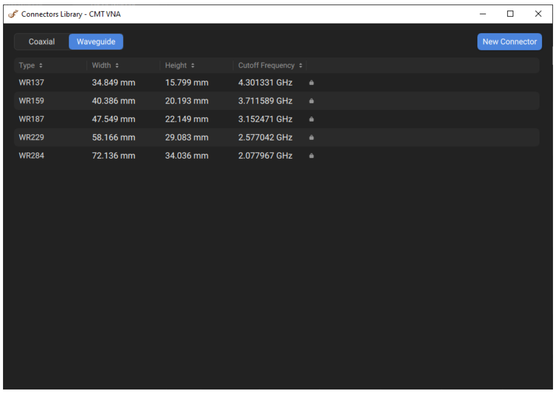
The Edit Connectors pop-up window displays a table of available connector types in the software. Connectors are grouped by the offset media type: coaxial and waveguide (see figures below).

Before measurements, ensure that the selected connector types match the connectors of the DUT.

By default, the table contains predefined connectors whose parameters cannot be modified. You can add a new user-defined connector. Editing and deleting functions are only available for user-defined connectors.



Coaxial Connectors Table



Waveguide Connectors Table

The tables below describe the columns in the connectors table.

Columns of the Coaxial Connector Table

Column	Description
Type	Connector type name.
Impedance	The characteristic impedance for the coaxial connector. Value has an impact on various parameter conversions. For example, the impedance enters into the calculation of the S-parameters for the calibration standards, if they are derived from a circuit model. The calculation of the (default) reference impedances for balanced ports.
Max Freqency	The maximum operating frequency of the coaxial connector, which is usually specified by the

Column	Description
	manufacturer with a margin relative to the maximum frequency.
Icon 	An icon representing the predefined connector.
Status **Genderless**	Indicates if a coaxial connector is genderless; this status is displayed in the corresponding row of the table.

Columns of the Waveguide Connector Table

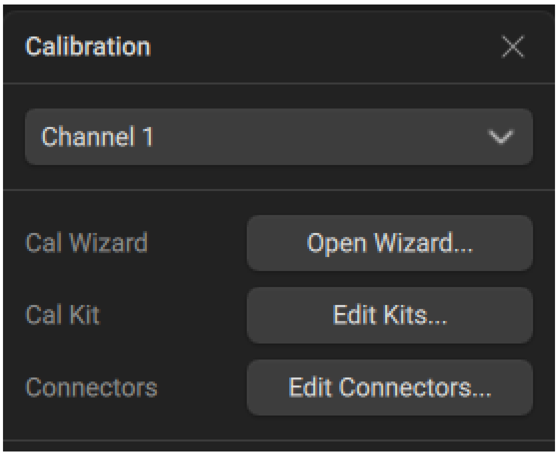
Column	Description
Type	Connector type name.
Width	The waveguide width.

Column	Description
Height	The waveguide height. Used in the waveguide loss model when the loss value is not zero.
Cutoff Frequency	The cutoff frequency of the waveguide. The cutoff frequency of the waveguide is achieved at a wavelength in the waveguide equal to twice its width. Note that this is different from the minimum and maximum operating frequency of the waveguide, which are usually specified by the manufacturer with a margin relative to the cutoff frequency.
Icon 	An icon representing the predefined connector.

Starting Connectors Pop-up Window

---

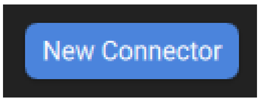
1. Click the **Calibration > Edit Connectors** buttons in the sidebar.



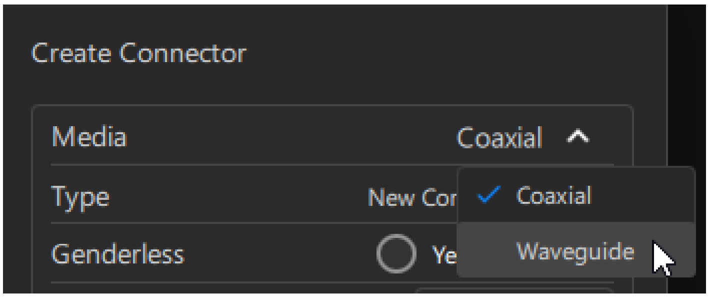
Adding New User-Defined Connector

New connector is added to the end of the table.

1. Click the **New Connector** button in the Connectors pop-up window.



2. Select media between **Coaxial** and **Waveguide** from the drop-down list in the **Media** field in the Create Connector pop-up window.



3. When **Coaxial** media is selected:

- Click the **Type** field and enter the connector type name.
- Adjust **Genderless**:

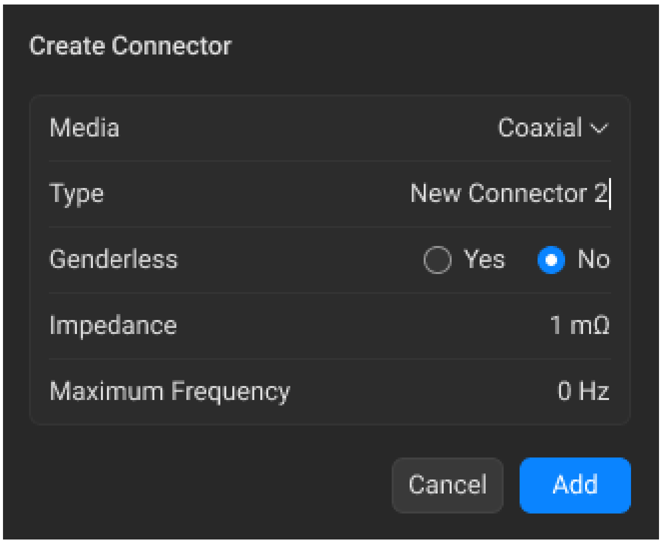
  If the connector is genderless, click the **Yes** radio button in the **Genderless** field.

  NOTE The corresponding status will appear to the right of the selected connector:

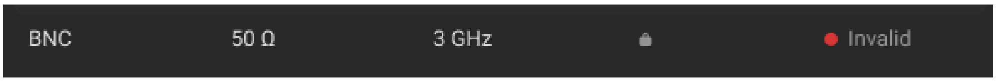
  If the connector is gendered, click the **No** radio button in the **Genderless** field (default).

  NOTE For the gendered connector, it is necessary to specify the gender (male/female) in the Standards pop-up window (see Standard Definition).

- Click the **Impedance** field and enter the impedance value.
- Click the **Maximum Frequency** field and enter the maximum frequency value at which the connector operates.

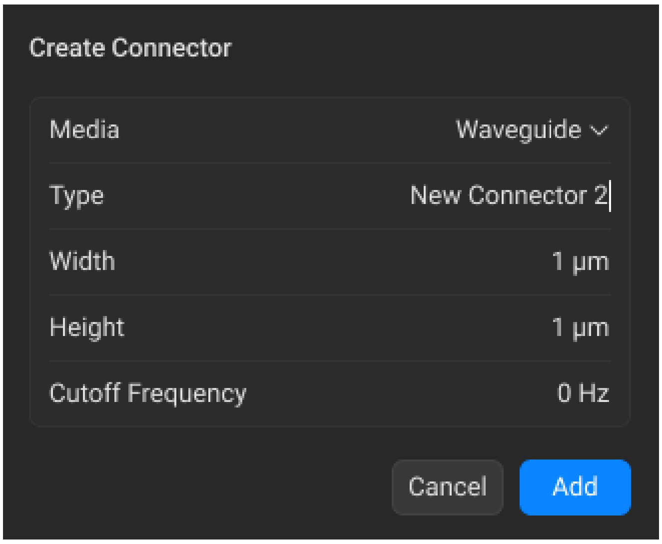


NOTE If zero values are entered for impedance or maximum frequency, the status "Invalid" appears in the corresponding row.



4 When **Waveguide** media is selected:

- Click the **Type** field and enter the connector type name.
- Click the **Width** field and enter the waveguide width value.
- Click the **Height** field and enter the waveguide height value.
- Click the **Cutoff Frequency** and enter the cutoff frequency value of the waveguide.

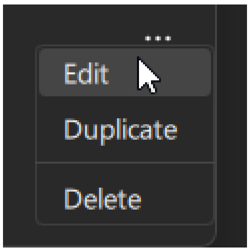


5 Click the **Add** button.

Editing User-Defined Connector

User-defined connectors can be edited. This function is not available for predefined connectors.

1. Click the horizontal-ellipsis button ··· next to the desired user-defined connector or right-click in the connector area, and select **Edit** in the dialog window.



2. When **Coaxial** media is selected:

- Click the **Type** field and edit the connector type name.
- Adjust **Genderless**:

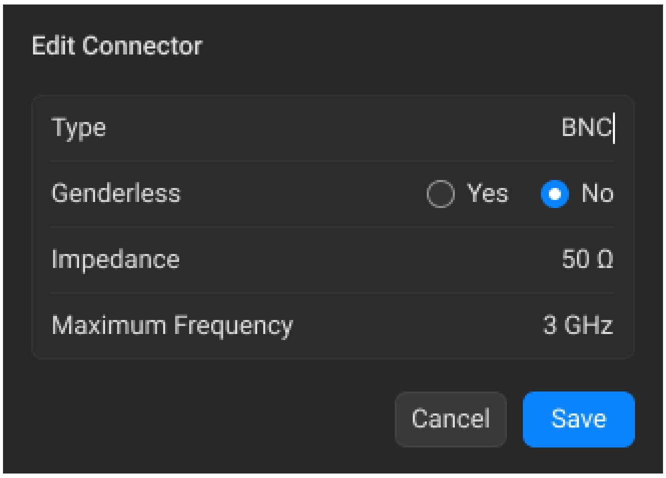
    If changing connector to genderless, select **Yes** in the **Genderless** field.

    NOTE The corresponding status will appear to the right of the selected connector:

    If changing connector to gendered, select **No** in the **Genderless** field.

    NOTE For the gendered connector, it is necessary to specify the gender (male/female) in the Standards pop-up window (see Standard Definition).

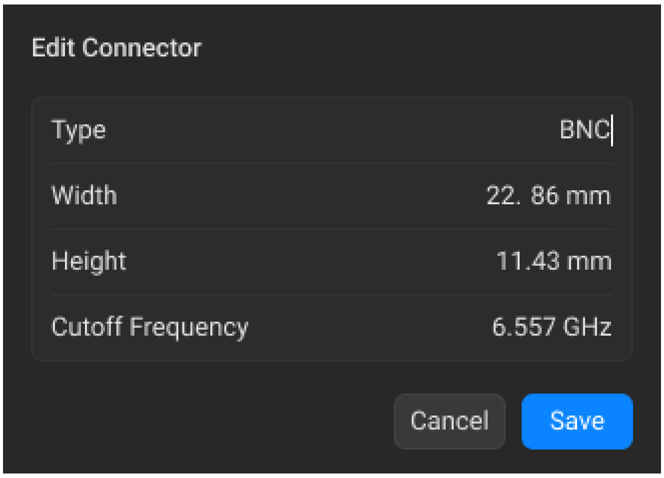
- Click the **Impedance** field and edit the impedance value.
- Click the **Maximum Frequency** field and edit the maximum frequency value at which the connector operates.



NOTE For the gendered connector, it is necessary to specify the gender (male/female) in the Standards pop-up window (see Standard Definition).

3 When **Waveguide** media is selected:

- Click the **Type** field and edit the connector type name.
- Click the **Width** field and edit the waveguide width value.
- Click the **Height** field and edit the waveguide height value.
- Click the **Cutoff Frequency** and edit the cutoff frequency value of the waveguide.

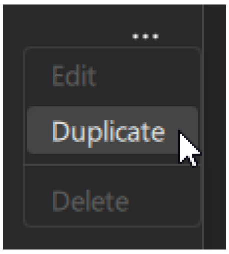


4 Click the **Save** button.

Duplicating Connectors

You can duplicate both predefined and user-defined connectors. In this case, duplicated predefined connector loses its status and can be modified, if necessary.

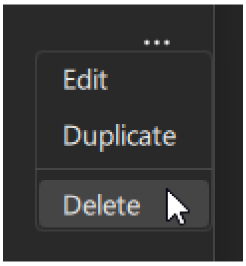
1. Click the horizontal-ellipsis button ··· next to the desired connector or right-click in the connector area, and select **Duplicate** in the dialog window.



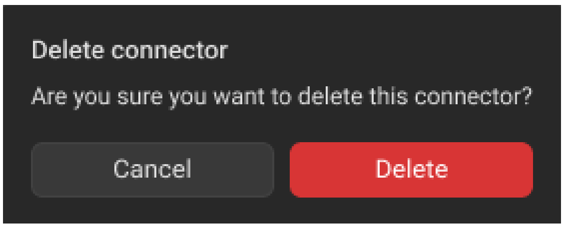
Deleting User-Defined Connector

User-defined connectors can be deleted. This function is not available for predefined connectors.

1. Click the horizontal-ellipsis button ··· next to the desired user-defined connector or right-click in the connector area, and select **Delete** in the dialog window.



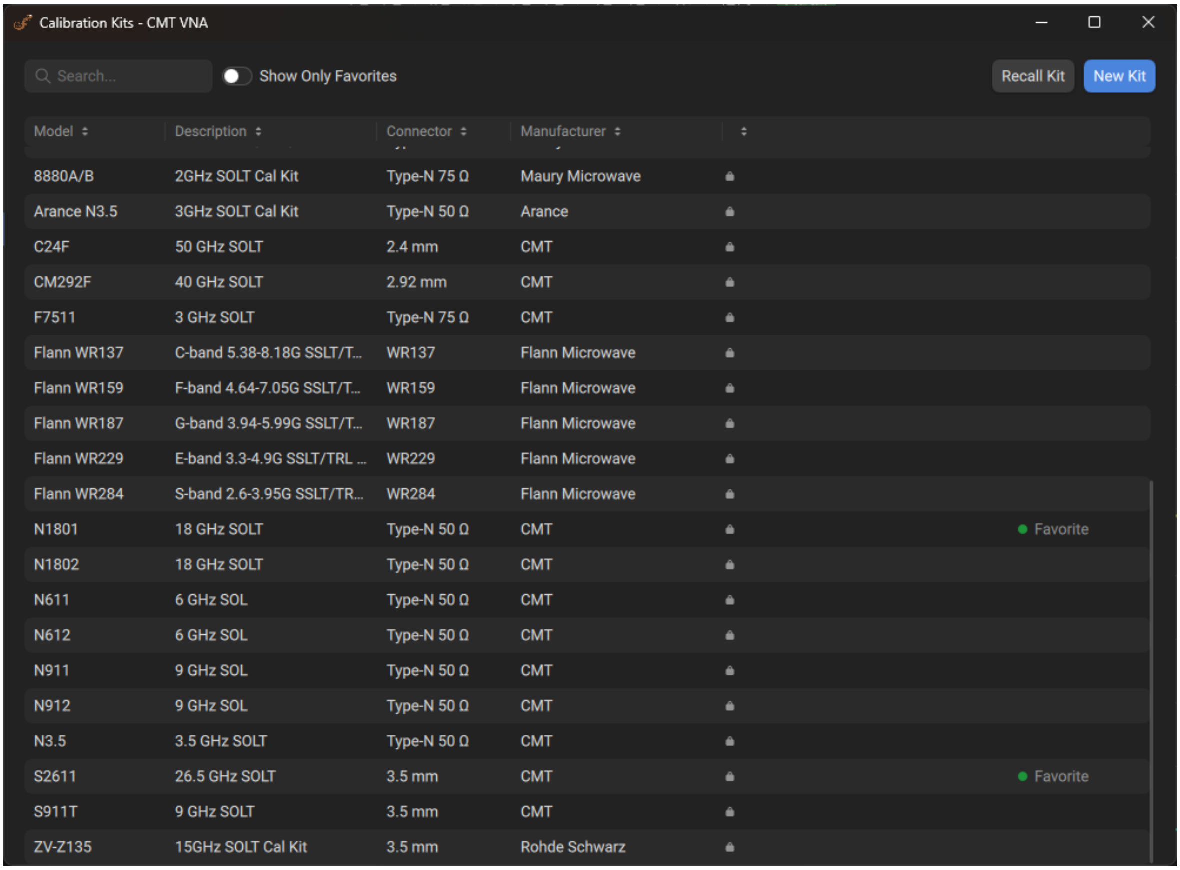
2. Click the **Delete** button in the opened dialog window.



NOTE If you delete the connector used in the User-Defined calibration kits, all kits will replace this type of connector with the first one in the table (N 50 Ω).

Calibration Kit Management

The Calibration Kits dialog window manages the calibration kit description. The window contains a table of calibration kits. Each row in the table describes one calibration kit. For the description of the columns from the table of calibration kits, see table below. You can add, duplicate, load or search for a calibration kit (see figure below).



Model	Description	Connector	Manufacturer		
8880A/B	2GHz SOLT Cal Kit	Type-N 75 Ω	Maury Microwave		
Arance N3.5	3GHz SOLT Cal Kit	Type-N 50 Ω	Arance		
C24F	50 GHz SOLT	2.4 mm	CMT		
CM292F	40 GHz SOLT	2.92 mm	CMT		
F7511	3 GHz SOLT	Type-N 75 Ω	CMT		
Flann WR137	C-band 5.38-8.18G SSLT/T...	WR137	Flann Microwave		
Flann WR159	F-band 4.64-7.05G SSLT/T...	WR159	Flann Microwave		
Flann WR187	G-band 3.94-5.99G SSLT/T...	WR187	Flann Microwave		
Flann WR229	E-band 3.3-4.9G SSLT/TRL ...	WR229	Flann Microwave		
Flann WR284	S-band 2.6-3.95G SSLT/TR...	WR284	Flann Microwave		
N1801	18 GHz SOLT	Type-N 50 Ω	CMT		Favorite
N1802	18 GHz SOLT	Type-N 50 Ω	CMT		
N611	6 GHz SOL	Type-N 50 Ω	CMT		
N612	6 GHz SOL	Type-N 50 Ω	CMT		
N911	9 GHz SOL	Type-N 50 Ω	CMT		
N912	9 GHz SOL	Type-N 50 Ω	CMT		
N3.5	3.5 GHz SOLT	Type-N 50 Ω	CMT		
S2611	26.5 GHz SOLT	3.5 mm	CMT		Favorite
S911T	9 GHz SOLT	3.5 mm	CMT		
ZV-Z135	15GHz SOLT Cal Kit	3.5 mm	Rohde Schwarz		

Calibration Kits Window

By default, the table contains predefined kits whose parameters cannot be modified.

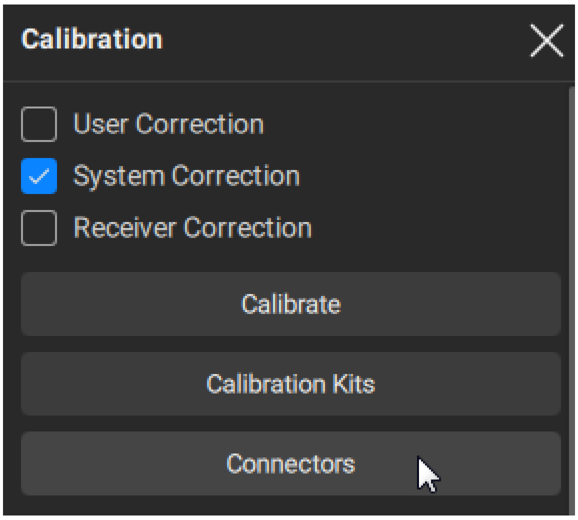
You can add user-defined calibration kits. A new user-defined kit can be added if a required kit is not included in the list of predefined kits. The user-defined calibration kits can be edited or deleted at any time.

Columns of the Calibration Kit Table

Column	Description
Model	The model of the calibration kit.
Description	Contains information about the type of standard and its characteristics within a specified frequency range.
Connector	The type of connectors of all standards in calibration kit (see Description of Connectors).
Manufacturer	Displays the manufacturer of the calibration kit.
Status **Empty**	Indicates that the calibration kit lacks description.
Status **Available**	Indicates that the calibration kit is available for selection in the Calibration Wizard. For instructions on setting the status, see Showing Only Available Kits. NOTE In the Calibration Wizard, it will be possible to select only calibration kits marked as Available. Kits without this status will not be shown in the wizard.
Icon 	An icon representing the predefined connector.

Starting Calibration Kits Pop-up Window

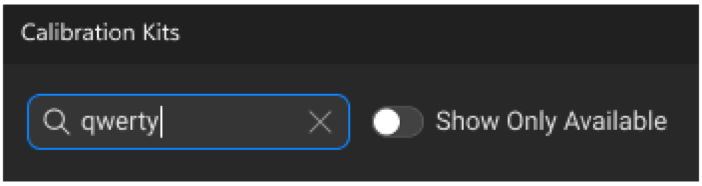
1 Click the **Calibration > Edit Kits** buttons in sidebar.



Searching Calibration Kit

You can use this option to search for a specific calibration kit from the list of kits saved in the software.

1. Click the **Search** field in the top-left area of the Calibration Kit dialog window and enter the desired calibration kit name.

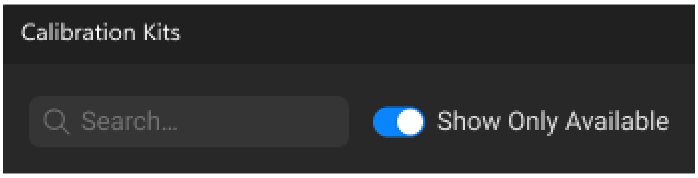


NOTE If no calibration kits are found, you can add a new user-defined calibration kit (see Adding New User-Defined Calibration Kit).

Showing Only Available Kits

This option hides unavailable calibration kits from the calibration kits table.

1. Toggle **Show Only Available** ON in the top-left area of the Calibration Kits dialog window.

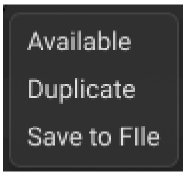


NOTE If no available calibration kits are found, you may need to adjust the status of kits manually (see Selecting Available or Unavailable Kits).

Selecting Available or Unavailable Kits

You can manually mark calibration kits as available or unavailable for selection in the Calibration Wizard.

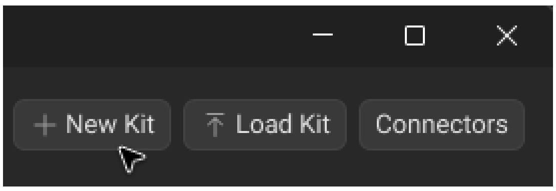
1. Click the horizontal-ellipsis button  next to the desired kit or right-click in the kit area, and select **Available** or **Unavailable** in the dialog window.



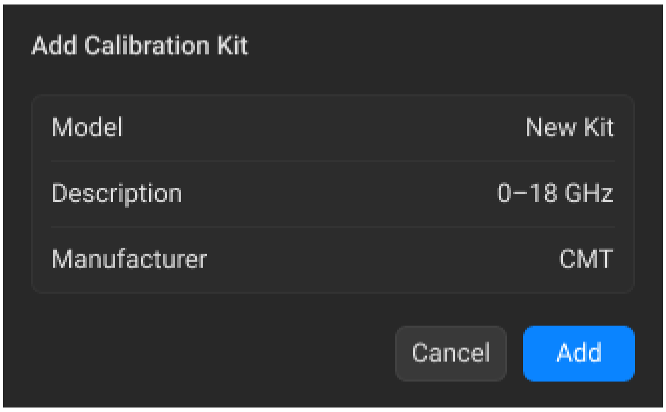
Adding New User-Defined Calibration Kit

New calibration kits are added to the end of the table.

1. Click the **New Kit** button in the top-right area of the Calibration Kits pop-up window.



2. Click the **Model**, **Description,** and **Manufacturer** fields in the **Add Calibration Kit** dialog window and enter the text.

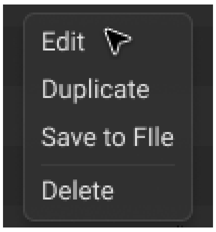


3 Click the **Add** button.

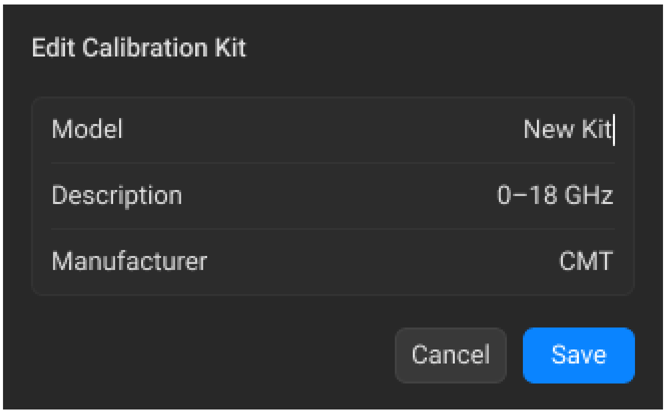
Editing User-Defined Calibration Kit Model, Description, and Manufacturer

The model of a calibration kit, its manufacturer, and description of the user-defined can be edited in the **Edit Calibration Kit** dialog window. The model appears in the Calibration Wizard pop-up window in the calibration kit selection field (see Selecting Calibration Plane). The manufacturer and description are informational fields.

1 Click the horizontal-ellipsis button ··· next to the desired kit or right-click in the kit area, and select **Edit** in the dialog window.



2 Click the **Model**, **Description,** and **Manufacturer** fields in the **Edit Calibration Kit** dialog window and enter the new text.



NOTE If the **Description** field is empty, the status "Empty" appears in the corresponding row.

3 Click the **Save** button.

Editing Calibration Kit Standards

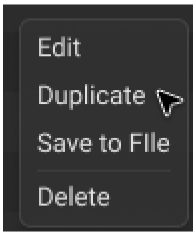
1 Click the **Connectors** button in the top-right area of the Calibration Kits dialog window or double-click the required row in the table of calibration kits.

Next, the Standards pop-up window opens. For a detail description of this window, see Standards Management in Calibration Kit.

Duplicating Calibration Kit

You can duplicate both predefined and user-defined calibration kits. When duplicating a predefined kit, the duplicated version loses its predefined status and can be modified, if necessary.

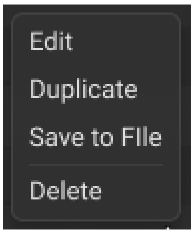
1. Click the horizontal-ellipsis button next to the desired kit or right-click in the kit area, and select **Duplicate** in the dialog window.



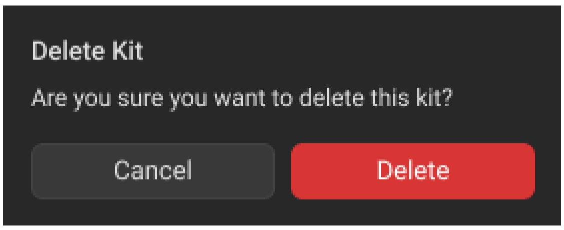
Deleting User-Defined Calibration Kit

User-defined calibration kits can be deleted. This function is not available for predefined kits.

1. Click the horizontal-ellipsis button next to the desired kit or right-click in the kit area, and select **Delete** in the dialog window.



2. Click the **Delete** button in the opened dialog window.



Saving Calibration Kits to File

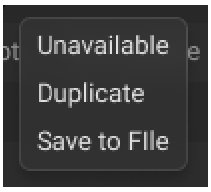
Saving a calibration kit to file is used for copying it to another line of the table or for transferring it to another Analyzer.

The calibration kits is saved as a *.NCK file on the hard drive and can be recalled later .

Changes made to the kit definition are saved automatically.

---

1. Click the horizontal-ellipsis button  next to the desired kit or right-click in the kit area, and select **Save to File** in the dialog window.



2. Select a path and enter the file name in the **Save calibration kit to file** dialog.

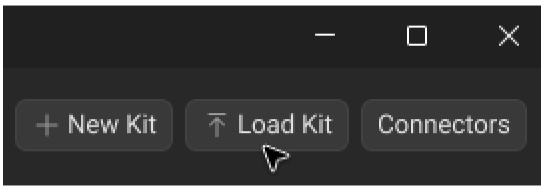
SCPI	MMEMory:STORe:CKIT

Loading Calibration Kits from File

Calibration kit files created by the previous command can also be loaded. Calibration kits from the file are added to the end of the table.

---

1. Click the **Load Kit** button in the top-right area of the Calibration Kits pop-up window.



2. Select a path and the file name in the **Load calibration kit from file** dialog.

SCPI	MMEMory:LOAD:CKIT

Standards Management in Calibration Kit

The **Standards** window provides detailed information about each standard included in the selected calibration kit (see figure below). This window contains the following areas:

Standards List – Displays a list of all standards available in the calibration kit.

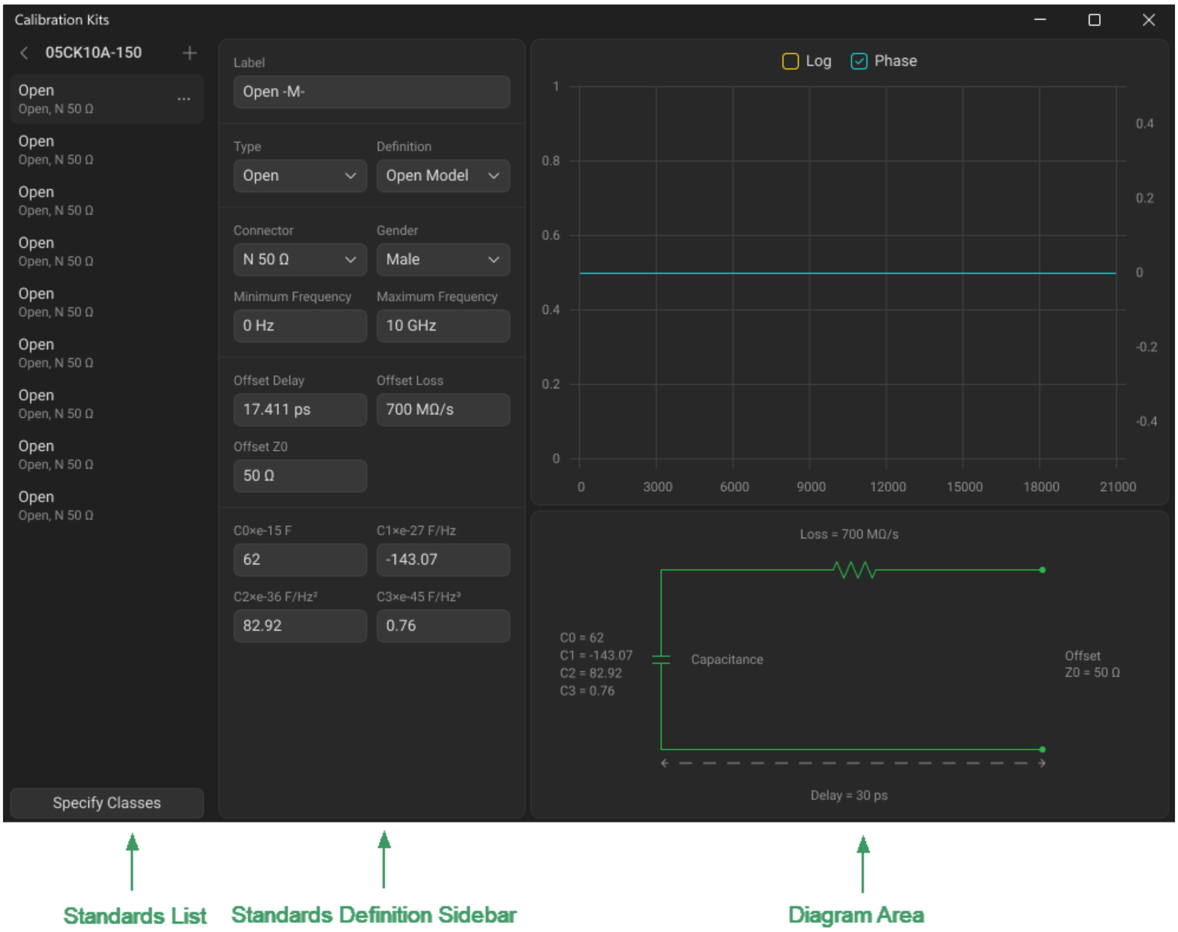
Standards Definition Sidebar – Provides options for defining each standard (see Standards Definition). For predefined kits, the parameters are view-only.

Diagram Area – This area automatically generates a graph based on the parameters in the Standards Definition Sidebar, displaying characteristics such as Log and Phase.

Equivalent Circuit Model (for circuit model standards only) – Located below the diagram, this area shows an equivalent circuit model. The values for capacitance, inductance, or arbitrary impedance are derived from the corresponding parameters in the Standards Definition Sidebar, providing a visual representation of the standard's circuit characteristics.

To open the **Standards** window, double-click on the desired calibration kit row in the **Calibration Kits** window (see Calibration Kit Management). The name of the active calibration kit is displayed in the title bar at the top of the window.

For user-defined calibration kits, you can add, edit, or delete standards. These functions are not available for predefined calibration kits.

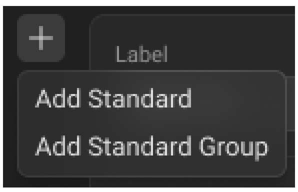


Standards Window

The instructions below cover managing standards in the **Standards List**. For detailed information about defining and editing standards, see Standards Definition for general information, Circuit Model Standards Definition for circuit model standards or Data-Based Standards Definition for data-based standards.

Adding New Standard to Calibration Kit

1. Click the **Add Standard** icon ＋ in the Standards list, then the **Add Standard** button. New standards are added to the end of the table.



Adding Standard Group to Calibration Kit

This function allows you to add a group of standards with specified connector type and frequency range. These parameters will apply to all standards added in this group.

---

1. Click the **Add Standard** icon  in the Standards list and select **Add Standard Group**.

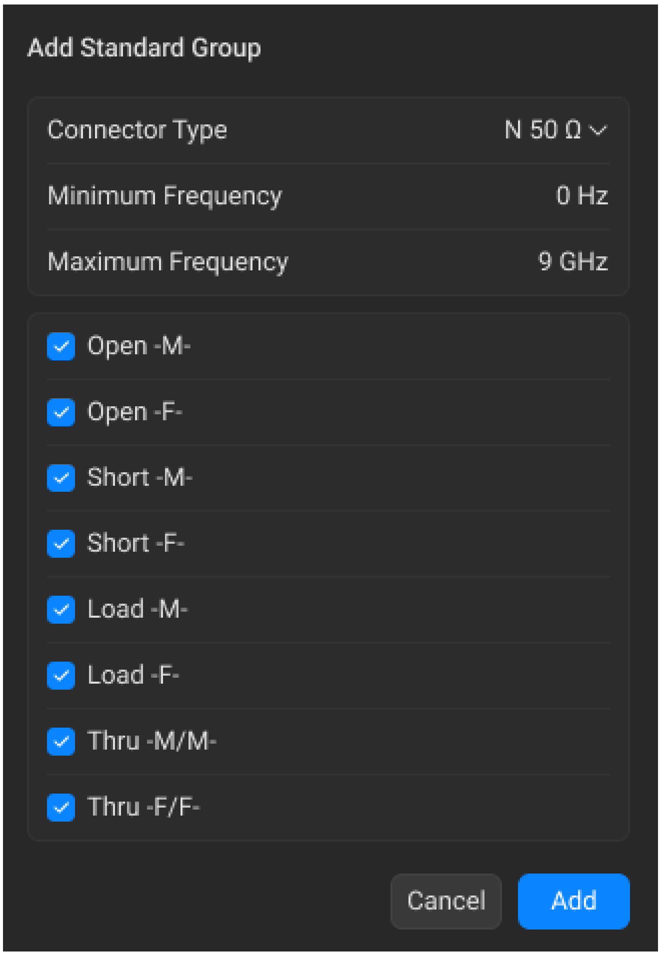
---

2. In the **Add Standard Group** dialogue window:

- Select the desired connector type from the **Connector Type** drop-down list.

NOTE The drop-down list contains connectors from the Connectors Library.

- Enter the frequency range in the **Minimum Frequency** and **Maximum Frequency** fields.



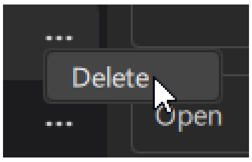
3. In the list of standards, use the checkboxes to select which standards to include in the calibration kit.

   NOTE All checkboxes are selected by default.

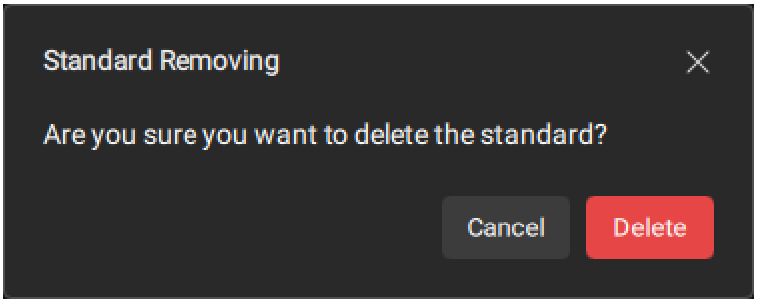
4. Click **Add**.

Deleting Standard from Calibration Kit

1. Click the horizontal-ellipsis button ··· next to the desired standard or right-click in the standard area, and select **Delete** in the dialog window.

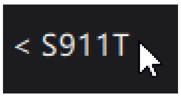


2. Click the **Delete** button in the opened dialog.



Back to Calibration Kits Management

1. Click on the calibration kit title at the top of the window.



Standards Definition

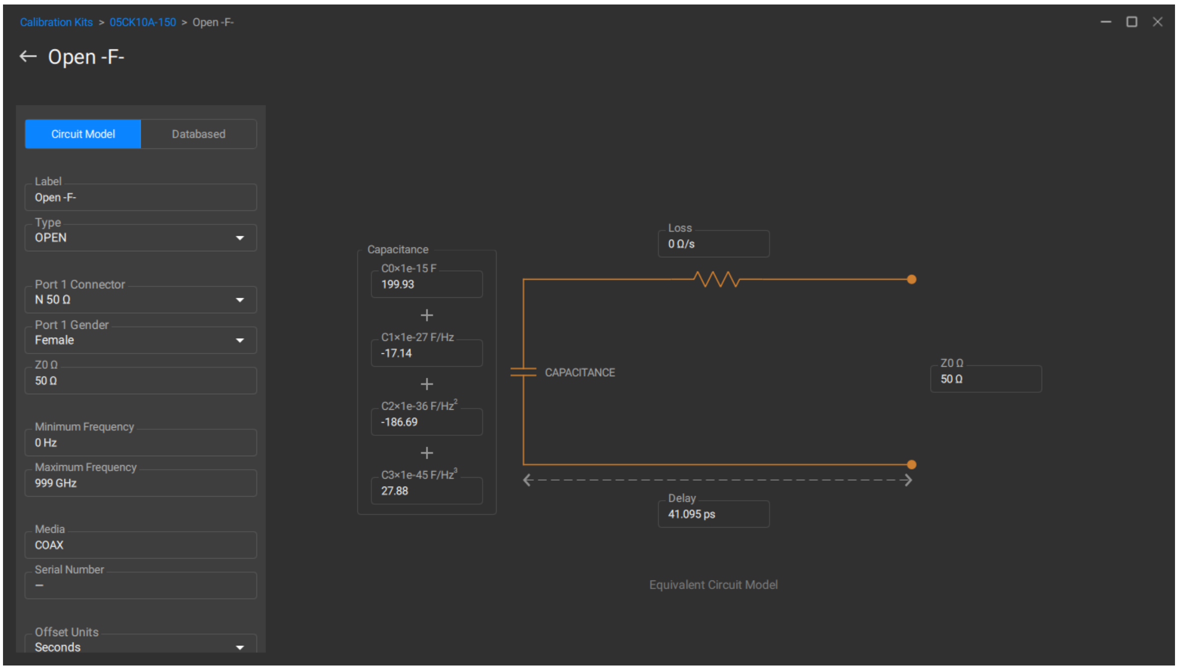
The Standards Definition sidebar provides the calibration kit standard definition. The Analyzer supports two methods for defining a standard:

- Circuit Model
- Data-Based (The calibration standards defined by S-parameters are called data-based standards).

Each standard is characterized by the lower and upper values of the operating frequency for the coaxial, or by the cutoff and double cutoff frequencies for the waveguide media.The measurements of the standards outside the specified frequency range are ignored in the process of calibration.

For detailed information about defining and editing standards, see Circuit Model Standards Definition for circuit model standards or Data-Based Standards Definition for data-based standards.

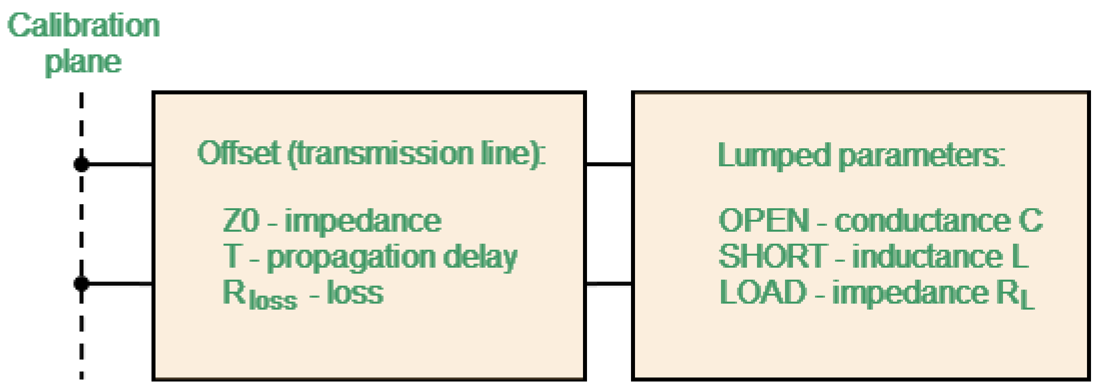
An example of the Standard Definition pop-up window defined by the circuit model for one calibration standard is shown below.



Circuit Model Standards Definition

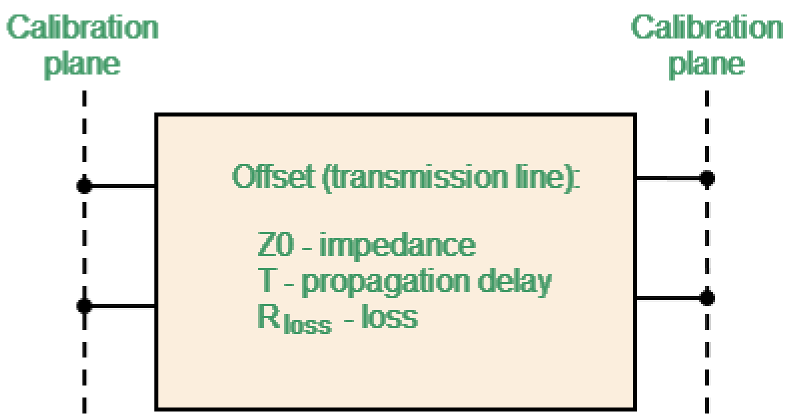
A model of a calibration standard presented as an equivalent circuit is used for determining the S-parameters of the standard. The model is employed for OPEN, SHORT, LOAD, THRU/DELAY standard types.

The one-port model is used for OPEN, SHORT and LOAD (see Full One-Port Calibration) standards. This is shown in the figure below.

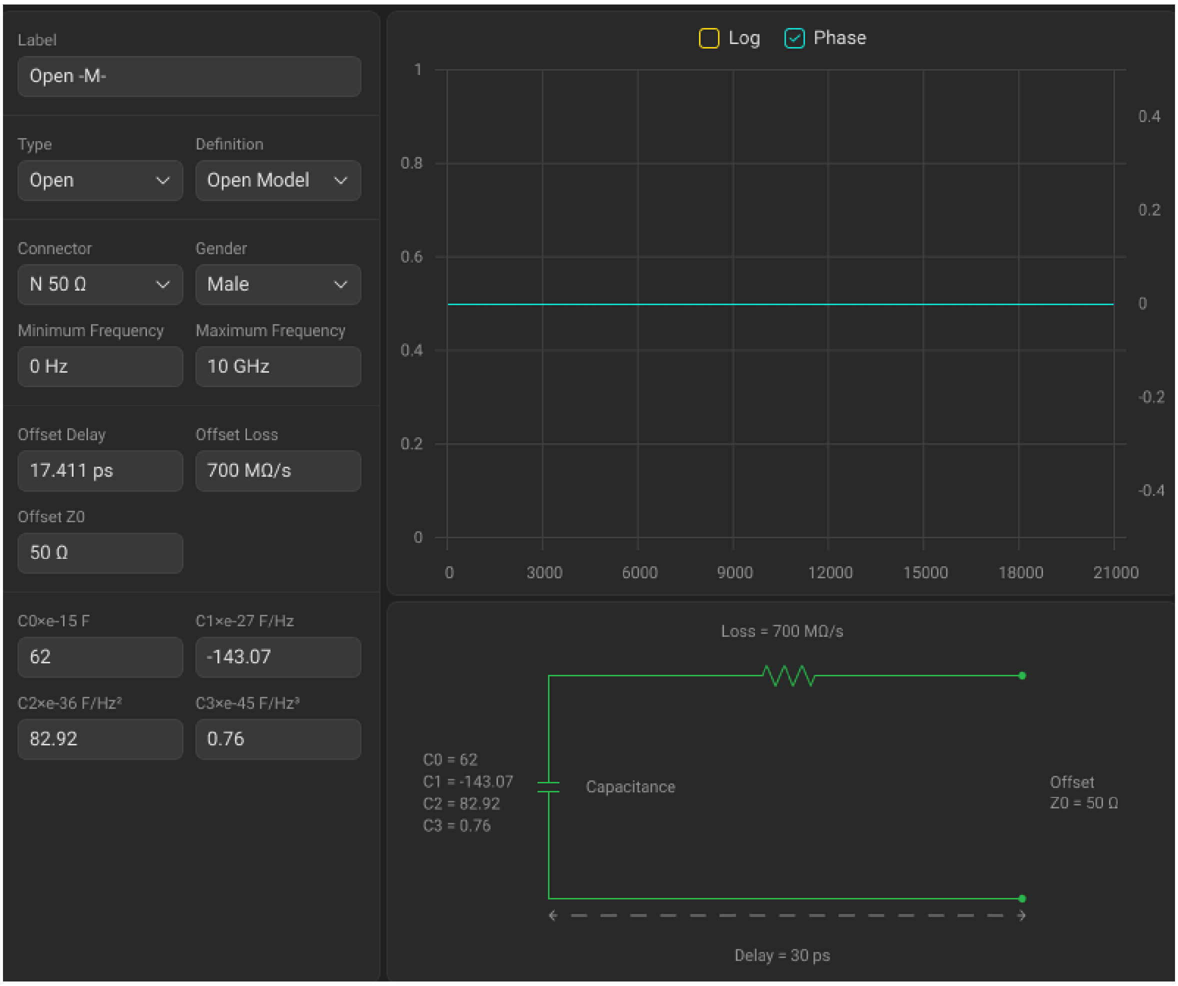


One-Port Standard Model

The two-port model is used for THRU and DELAY standards (see figure below).



Two-Port Standard Model



Standards Definition Sidebar for the Standards Defined by the Circuit Model

The description of the Standards Definition Sidebar of a calibration standard defined by an equivalent circuit model is shown in the table below.

Field (as in the software)	Field Definition
Label	Standard label
Type	The standard type (see Types of Calibration Standards)
Definition	The Analyzer supports two methods for defining a standard: • Circuit Model

Field (as in the software)	Field Definition
	• Databased (The calibration standards defined by S-parameters are called Databased standards).
Connector	The type of standard connector (see Editing User-Defined Connector Type)
Gender	The gender of a coaxial standard: • Male • Female
Minimum Frequency, Maximum Frequency	The minimum and maximum standard operating frequency in the coaxial. Used for calibration that uses several calibration standards, each of which does not cover entire frequency range.The measurements of the standards outside the specified frequency range are ignored in the process of calibration.
Width, Height	The waveguide width and height. Used in the waveguide loss model when the loss value is not zero.
Offset Delay	The offset delay. It is defined as one-way signal propagation time in the transmission line. Unit: [seconds]. The delay can be measured or mathematically determined by dividing the exact physical length by the propagation velocity in the line. For waveguide, delay is conventionally taken to be equal to the delay of a coaxial line of the same length. The actual signal delay in waveguide is frequency-dependent and is calculated in the software. Instead of delay, you can specify the length of the offset. Unit: [meters]. The software calculates the delay according to the formula for a coaxial air line: $T = \frac{\sqrt{\varepsilon_r} l}{c}$,

Field (as in the software)	Field Definition
	where l — line length [m], c — light speed in free space 299792458 [m/s], ε_r— relative permittivity of air 1.000649. The offset delay can be switched to the physical length [m] (see Selecting Offset Units). The length can be specified instead of the delay, provided that the offset of the calibration standard is a coaxial airline or a waveguide. If the calibration standard manufacturer provides delay data, specify delay.
Offset Loss	The offset loss in one-way propagation due to the skin effect. Unit: [Ω/sec]: • The loss in the coaxial transmission line is determined by measuring the delay T [sec] and loss L [dB] at a frequency of 1 GHz. The measured values are used in the following formula: $R\pi[\Omega/s] = \frac{L[dB] \cdot Z0[\Omega]}{4.3429[dB] \cdot T[s]}$ • The loss in waveguide is typically set to 0 due to its very small influence. However, the software supports a waveguide loss model. If the calibration standard manufacturer provides loss data, specify it.
Offset Z0	The characteristic impedance of the transmission line, serving as the offset. Unit: [Ω]. For the coaxial line specified real value of characteristic impedance, usually equal to 50 Ω or 75 Ω. NOTE Z0 of the selected connector type is installed by default.
Capacitance **(C0, C1, C2, C3)**	The fringe capacitance of an OPEN standard, which causes a phase offset of the reflection coefficient at high frequencies. The fringe capacitance model is

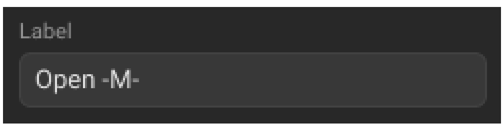
Field (as in the software)	Field Definition
	described as a function of frequency, which is a polynomial of the third degree: $C = C0 + C1 \cdot f + C2 \cdot f^2 + C3 \cdot f^3$, where f — frequency [Hz], C0…C3 — polynomial coefficients. Units: C0[F], C1[F/Hz], C2[F/Hz^2], C3[F/Hz^3].
Inductance **(L0, L1, L2, L3)**	The residual inductance of a SHORT standard, which causes a phase offset of the reflection coefficient at high frequencies. The residual inductance model is described as a function of frequency, which is a polynomial of the third degree: $L = L0 + L1 \cdot f + L2 \cdot f^2 + L3 \cdot f^3$, where f — frequency [Hz], L0…L3 — polynomial coefficients. Units: L0[H], L1[H/Hz], L2[H/Hz^2], L3[H/Hz^3].
Arbitrary Impedance	Load impedance of a fixed load calibration standard. Unit: [Ω]. For the coaxial calibration standard specified real value of characteristic impedance, usually equal to 50 Ω or 75 Ω.

Editing Standard Label

1. Select the desired standard in the Standards List by clicking on it.

   NOTE If there are no suitable standards, you can add a new standard (see Adding New Standard to Calibration Kit).

2. Click the **Label** textbox in the Standards Definition Sidebar and enter the standard label.



SCPI SENSe:CORRection:COLLect:CKIT:STAN:LABel

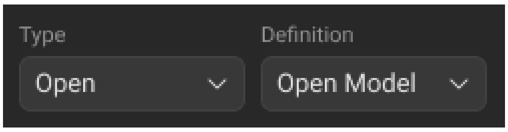
NOTE It is recommended to mark gender in Label textbox (see Gender of Calibration Standard).

Selecting Standard Type and Circuit Model Definition

1. Click the **Type** drop-down list in the Standards Definition Sidebar, and select standard type.

2. Click the **Definition** drop-down list in the Standards Definition Sidebar, and select:

   - **Open Model** for **Open** standard type.
   - **Short Model** for **Short** standard type.
   - **Matched**, **Sliding Load**, or **Arbitrary** for **Load** standard type.
   - **Thru Model** for **Thru** standard type.
   - **Line** for **Line** standard type.



SCPI SENSe:CORRection:COLLect:CKIT:STAN:TYPE

Selecting Standard Port Connector and Gender

1. Click the **Connector** (**Connector 1** and **Connector 2** for two-port standard only) drop-down list in the Standards Definition Sidebar, and select connector type.

   NOTE The drop-down list contains connectors from the Connectors Library.

2. Click the **Gender** (**Gender 1** and **Gender 2** for two-port standard only) drop-down list in the Standards Definition Sidebar, and select connector gender.

NOTE For coaxial gendered media only.

Editing Standard Minimum Frequency and Maximum Frequency

1. Click the required textbox in the Standards Definition Sidebar, and enter the numerical value.

   NOTE Editing minimum frequency and maximum frequency of the standard is usually not required, because it is set according to the selected connector.

SCPI SENSe:CORRection:COLLect:CKIT:STAN:FMINimum,
SENSe:CORRection:COLLect:CKIT:STAN:FMAXimum

Editing Waveguide Standard Width and Height

1. Click the **Width** and **Height** textboxes in the Standards Definition Sidebar, and enter the numerical value.

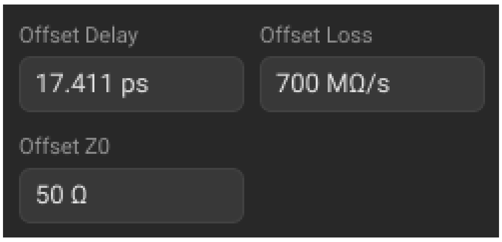
   NOTE For standards with waveguide media only. Editing width and height of the standard is usually not required, because it is set according to the selected connector.

Editing Offset Delay, Offset Loss, Offset Z0

1. Click the **Offset Delay** and **Offset Loss** textboxes in the Standards Definition Sidebar, and enter the numerical value.

2 Click the **Offset Z0** textbox in the Standards Definition Sidebar, and enter the numerical value.

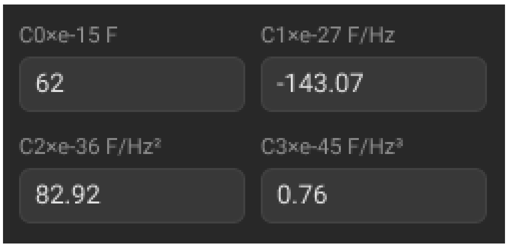
NOTE For coaxial media only. Editing offset Z0 impedance of the standard is usually not required, because it is set according to the selected connector.



SCPI SENSe:CORRection:COLLect:CKIT:STAN:Z0

Editing Standard Capacitance, Inductance, or Arbitrary Impedance

1 Click the required textbox in the Standards Definition Sidebar and enter the numerical value.



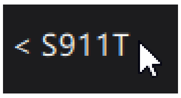
SCPI SENSe:CORRection:COLLect:CKIT:STAN:C0,
SENSe:CORRection:COLLect:CKIT:STAN:C1,
SENSe:CORRection:COLLect:CKIT:STAN:C2,
SENSe:CORRection:COLLect:CKIT:STAN:C3

SENSe:CORRection:COLLect:CKIT:STAN:L0,
SENSe:CORRection:COLLect:CKIT:STAN:L1,
SENSe:CORRection:COLLect:CKIT:STAN:L2,
SENSe:CORRection:COLLect:CKIT:STAN:L3

SENSe:CORRection:COLLect:CKIT:STAN:ARBitrary

Back to Calibration Kits Management

1 Click on the calibration kit title at the top of the window.

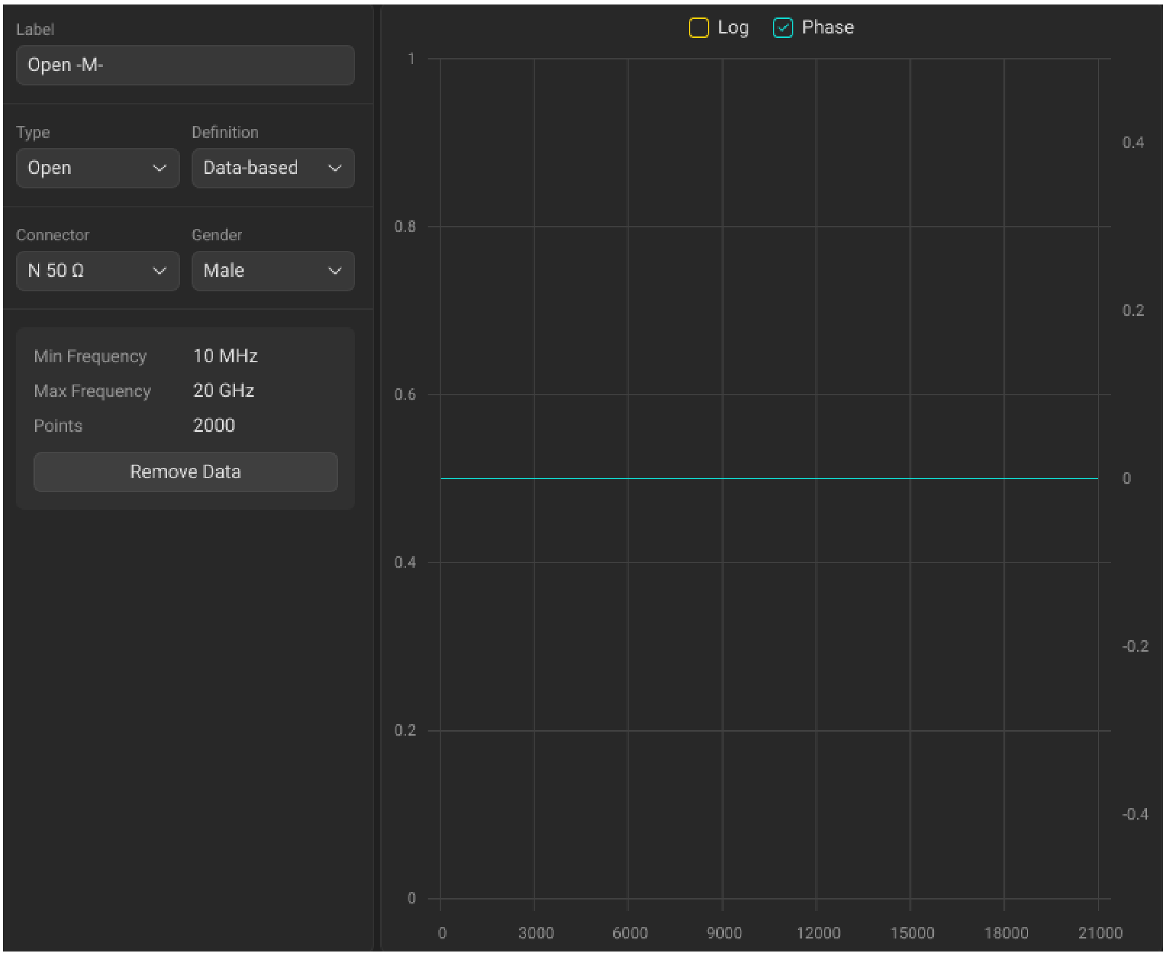


Data-Based Standards Definition

The calibration standards defined by data are set using the table of S-parameters. Each row of the table contains the frequency and S-parameters of the calibration standard. For one-port standards, the table contains the value of only one parameter — S11, and for two-port standards the table contains the values of all the four parameters — S11, S21, S12, S22.

S-parameters data can be downloaded from a Touchstone file. Files with a *.S1P extension are used for one-port standards, and files with a *.S2P extension are used for two-port standards.

The following rule is applied for the calibration of a two-port standard: the standard is considered to be connected by Port 1 (S11) to the port with the smallest number and by Port 2 (S22) to the port with the biggest number. If a two-port standard needs to be reversed, use the Reverse Port function.



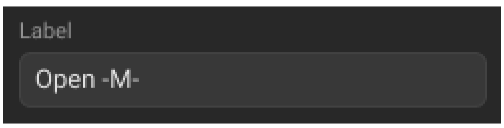
Standards Definition Sidebar for the Standards Defined by Data

The description of the Standards Definition Sidebar of a calibration standard defined by data is shown in the table below.

Field (as in the software)	Field Definition
Label	The name used to identify the standard.
Type	The standard type (see Types of Calibration Standards)
Definition	The Analyzer supports two methods for defining a standard: • Circuit Model • Data-Based (The calibration standards defined by S-parameters are called Databased standards).
Connector	The type of standard connector (see Editing User-Defined Connector Type)
Gender	The gender of a coaxial standard: • Male • Female
Data From the S-Parameters File	
Minimum Frequency, Maximum Frequency	The minimum and maximum operating frequencies for the standard. These values are derived from the loaded S-parameters and cannot be edited in the software.
Points	The number of frequency points, as determined by the S-parameter file.

Editing Standard Label

1. Click the **Label** textbox in the Standards Definition Sidebar and enter the standard label.



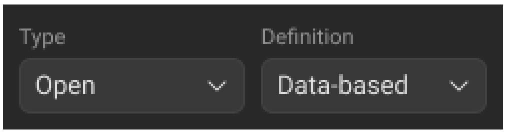
SCPI	SENSe:CORRection:COLLect:CKIT:STAN:LABel

NOTE	It is recommended to mark gender in Label textbox (see Gender of Calibration Standard).

Selecting Standard Type and Data-Based Definition

1. Click the **Type** drop-down list in the Standards Definition Sidebar, and select standard type.

2. Click the **Definition** drop-down list in the Standards Definition Sidebar, and select **Data-based**.



SCPI	SENSe:CORRection:COLLect:CKIT:STAN:TYPE

Selecting Standard Port Connector and Gender

1. Click the **Connector** (**Connector 1** and **Connector 2** for two-port standard only) drop-down list in the Standards Definition Sidebar, and select connector type.

   NOTE The drop-down list contains connectors from the Connectors Library.

2. Click the **Gender** (**Gender 1** and **Gender 2** for two-port standard only) drop-down list in the Standards Definition Sidebar, and select connector gender.

   NOTE For coaxial gendered media only.

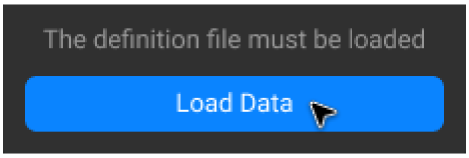
Loading Data from Touchstone File

The table of S-parameters can be loaded from Touchstone file. The Touchstone file type (*.S1P or *.S2P) is determined automatically depending on the type of standard.

---

1. Select standard type (see Selecting Standard Type and Data-Based Definition).

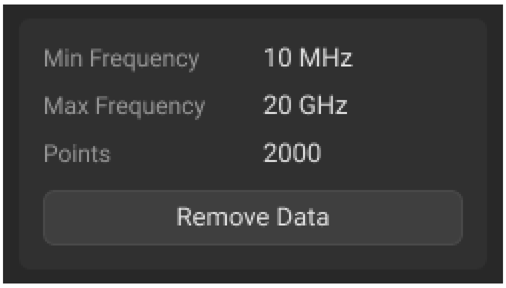
2. Click the **Load Data** button.



3. Select a path and the file name in the dialog window.

Deleting Touchstone File Data

2. Click the **Remove Data** button in the Touchstone data area.

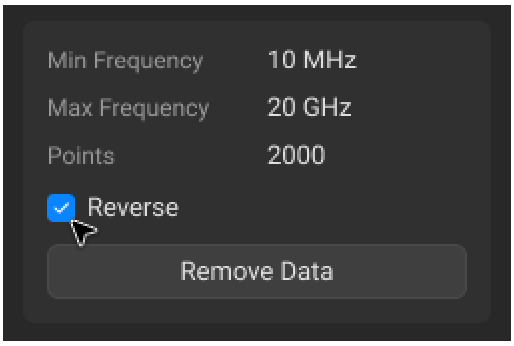


Reverse Port

Reverse Port is available for two-port standard. By default, the standard is considered to be connected by Port 1 (S11) to the port with the smallest number and by Port 2 (S22) to the port with the largest number. Use Reverse Port function if a two-port standard needs to be reversed.

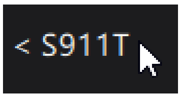
---

1. Select the **Reverse** checkbox in the Touchstone data area.



Back to Calibration Kits Management

1. Click the calibration kit title at the top of the window.



Classes of Calibration Standards

Along with defining a calibration standard by a calibration model or data, the standard should also be assigned a specific class. One calibration standard may belong to one class This class assignment is crucial for specifying properties such as the calibration method and the role of the standard in calibration.

To simplify the calibration process and guide users in selecting the correct standards, calibration families organize standards into logical groups, aligning with the steps necessary for accurate S-parameter measurements.

The two main families of calibration classes—**SOLT** and **TRL**—define which standards are required at each calibration step, ensuring consistency and reducing errors for a smoother and more reliable calibration process.

SOLT Family Classes

The SOLT family is based on the use of Short, Open, Load, and Thru standards, commonly used in traditional calibration methods.

In SOLT calibration, the process typically involves two one-port calibrations followed by a thru measurement. The thru connection may include either a defined thru (with a known definition) or unknown thru (with no predefined definition). The simplest definition of a thru standard is a flush thru, an ideal zero-length connection between two analyzer ports.

The following table lists the SOLT family classes and the recommended settings for each calibration method.

Calibration Methods	SOLT Class Label	Default Settings	Recommended Settings
Full N-Port Calibration Full One-Port Calibration One-Path N-Port Calibration Transmission Normalization Reflection Normalization	Reflection Std 1	Short	Short with minimum length (L1)
	Reflection Std 2	Open	Open and Short with length L2 > L1
	Reflection Std 3	Load	Load and Short with length L3 > L2

Calibration Methods	SOLT Class Label	Default Settings	Recommended Settings
	Thru Connection	Flush thru	Flush thru, Unknown thru, or any two-port standard with known S-parameters

TRL Family Classes

The TRL family is based on Thru, Reflect, and Line standards, commonly used in more advanced calibration scenarios, especially at higher frequencies.

TRL calibration requires measuring two two-port standards with identical structures but different electrical lengths, along with a high-reflection standard. Unlike SOLT, TRL operates over a frequency range where the phase difference between the thru and line standards is at least 20 degrees.

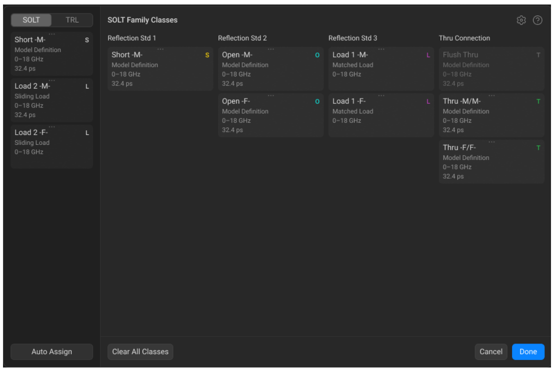
The following table lists the TRL family classes and the recommended settings for each calibration method.

Calibration Methods	SOLT Class Label	Default Settings	Recommended Settings
TRL/LRL/TRM/LRM Calibration	Thru/Line	Flush thru	Flush thru, or Line with the length L2 < L1
	Matched Load	Matched Load	Matched Load
	Line	Line	Line with the maximum length (L1)
	Reflection	Open	Open and Short

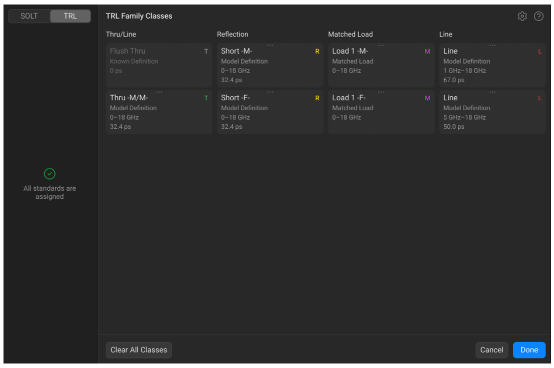
Classes Management

This section provides a detailed instructions for assigning classes to standards in a calibration kit.

The **Classes** window contains a list of standards that have been added to the selected calibration kit (see figure below). For a detailed information on how to add standards to a kit, see [Standards Management in Calibration Kit](). Each standard is marked with a letter that corresponds to its type or class within the selected calibration family. These standards need to be assigned to classes (e.g., **Reflection Std 1, Reflection Std 2**, etc.) within the **Classes Table** to form the calibration settings. One calibration standard may belong to single class in one family, with the assignment performed individually for each calibration kit.



SOLT Family Classes Window



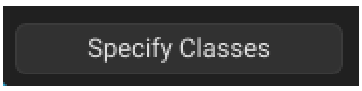
TRL Family Classes Window

The procedure for assigning classes to calibration standards is described below.

Accessing the Classes Window

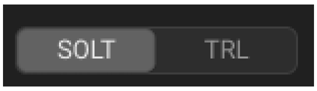
1. Select the calibration kit by double-clicking the required row in the Calibration Kits window (see [Calibration Kit Management]()).

2. Click the **Specify Classes** button at the bottom of the Standards List in the Standards Window.

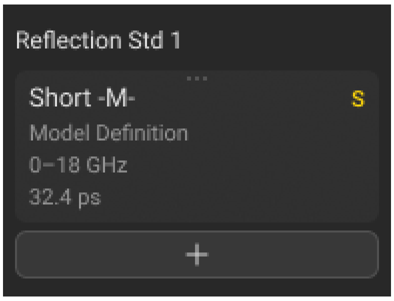


Assigning Classes to Standards

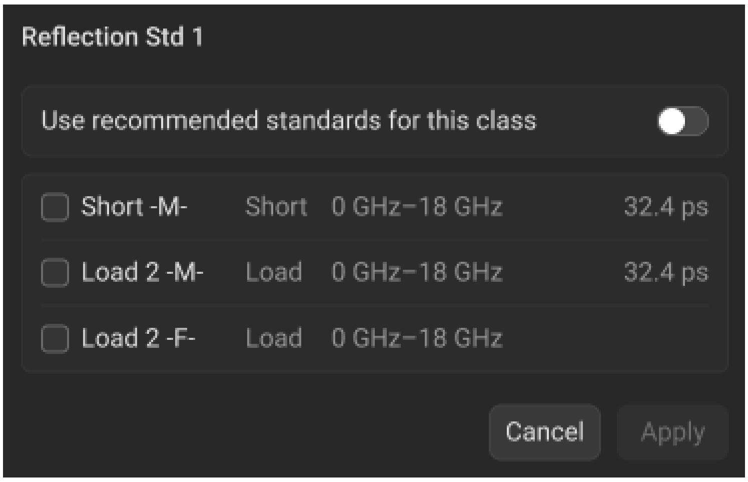
1. Click the **SOLT** or **TRL** segment button, depending on the calibration family you wish to configure.



2. Hover over the column for the desired class, and click the button that appears.



3. In the dialog window that appears, select the checkboxes for the standards you wish to add to the class.



NOTE To use the recommended standards for the class, toggle **Use recommended standards for this class** ON. For the list of recommended standards for each family, see the SOLT Family Classes table or the TRL Family Classes table.

4. Click **Apply**.

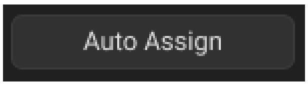
NOTE

You can also assign a standard to a class by dragging and dropping it into the corresponding column.

Auto Assigning

This function automatically allocates standards to classes within a calibration kit.

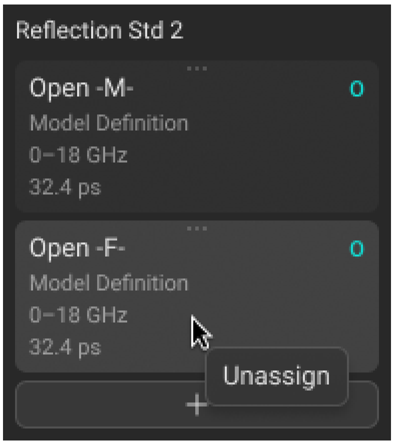
1. Click the **Auto Assign** button located at the bottom of the Classes window.



Unassigning Standards From Classes

This function allows you to unassign a standard from a class.

1. Right-click on the required standard in the class column, and then click the **Unassign** button.



Configuring standards for SOLT and TRL Calibration

This function allows to adjust specific settings for SOLT and TRL calibration standards.

1. Click the icon in the upper-right corner of the Classes window to open the dialog for configuring standards for SOLT and TRL calibration.

2. In the dialog that opens, configure the following options as needed:

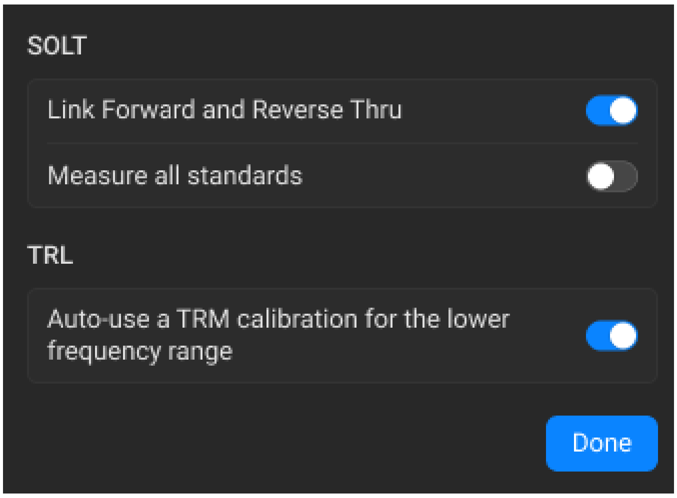
   SOLT Settings:

   Toggle the **Link Forward and Reverse Thru** ON to link the forward and reverse thru connections during calibration, simplifying the setups if they are identical.

   Toggle the **Measure all standards** ON to measure all available standards in the calibration kit.

   TRL Settings:

   Toggle the **Auto-use a TRM calibration for the lower frequency range** ON to automatically apply a TRM calibration for lower frequencies.

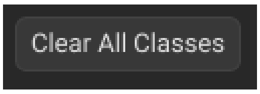


3 Click **Done**.

Clearing All Standards from the Standards Class Table

This function resets the class assignment.

1 Click the **Clear All Classes** button to delete all the standards in the table.



Subclasses of Calibration Standards

Subclasses are used to assign one class to several calibration standards. The procedure of subclass assignment is mainly employed for calibration within a wide frequency range by several calibration standards, each of which does not cover the full frequency range. Each class of standards can contain up to 8 subclasses. The procedure of subclass assignment to the calibration standards is described in Classes of Calibration Standards.

For example, suppose the LOAD standard is defined as from 0 to 2 GHz, and the sliding LOAD standard is defined as from 1.5 to 12 GHz. To perform calibration within the full frequency range, the fixed LOAD should be assigned the subclass 1, and the sliding LOAD should be assigned the subclass 2 of the “load" class.

If the standards have an overlapping frequency range (as in the example above, from 1.5 to 2 GHz), the last measured standard will be used.

NOTE Subclass assignment changes the labels of the calibration softkeys. The measurement softkey is replaced by the key, which opens the subclass menu containing the keys for measuring several calibration standards.

Calibration Wizard Overview

The Calibration Wizard allows to create and customize various calibration configurations. It is possible to create multiple configurations and perform calibration later by simply selecting the desired configuration from the list.

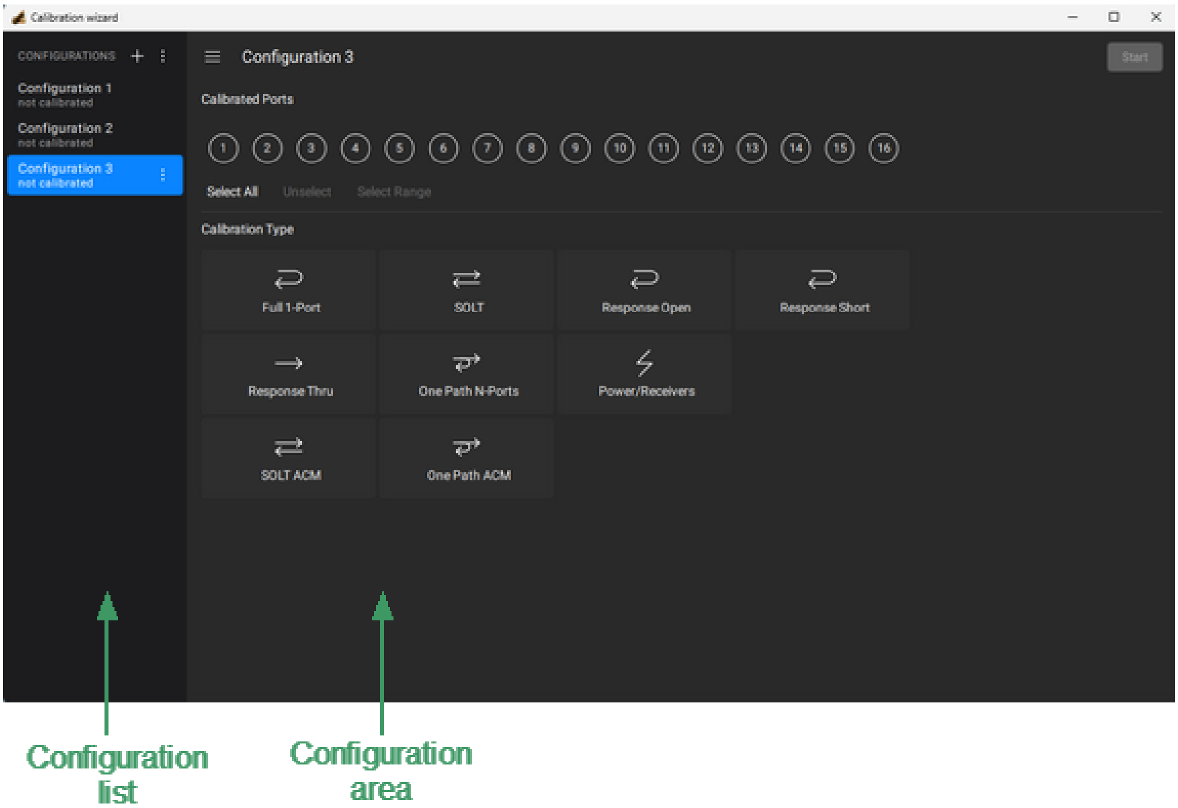
The configuration is used to combine ports into a group and assign a calibration type to that group. You can create several port groups in a single configuration and assign the required calibration types to them. For example, Port 1 can be combined into group 1. This group can be assigned to the Reflection Normalization (Response Open) calibration type. Then, Port 2 and Port 3 can be combined into group 2, and this group can be assigned Full one-port (SOL) calibration type. Next, Port 4, Port 5, Port 6 and Port 7 can be combined into group 3, and this group can be assigned another type of ACM calibration and so on.

Once ports have been grouped together, set the calibration plane parameters for all groups by selecting the connector type and connector gender for each port, as well as the calibration kit or ACM used in calibration. Then, perform the calibration step-by-step.

NOTE	Each port can only be used in one group. If necessary, ports can be rearranged.

NOTE	The Calibration Wizard does not allow to create S-parameter calibrations and power/receiver calibrations in the same configuration. Power/receiver calibrations should be done in a separate configuration.

The Calibration Wizard pop-up window is shown in the figure below. The window contains a configuration list with the created configurations and a configuration area for customization of calibration configuration. You can click on areas or labels in the figure to go to a description of the wizard parts.



Calibration Wizard Pop-up Window

The sequence of actions to perform calibration in the wizard:

1. Run Calibration Wizard (see Starting Calibration Wizard Pop-up Window).
2. Create a configuration (see Adding New Configuration) or use an existing configuration (see Editing a Configuration).
3. If creating a new configuration: create port groups and assign each group the required calibration type (see Creating a New Ports Group).
4. If creating a new configuration: select a calibration plane for each port involved in the calibration (see Selecting Calibration Plane). Calibration plane must be selected in accordance with the connectors of the measured DUT.
5. Run the configuration (see Starting Configuration).
6. Complete calibration steps.
7. Apply a configuration (see Applying a Configuration).

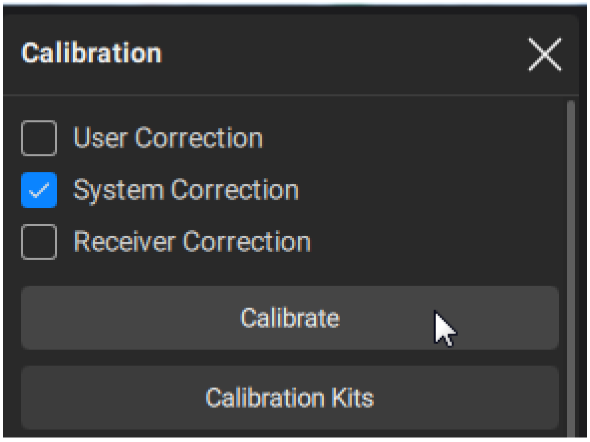
For a detailed description of each calibration type and step-by-step execution of the calibration procedure, see Calibrations Step-by-Step.

Starting Calibration Wizard Pop-up Window

1. Select the channel (see Selection of Active Trace, Diagram, Channel) and set the parameters of the channel (frequency range, IF bandwidth, etc).

2. If calibration is performed using the calibration kit, make sure required calibration kit is marked as **AVAILABLE** in Calibration Kits Management. If the calibration kit is missing, add the kit and its description.

3. Click on the **Calibration > Calibrate** buttons in the sidebar.

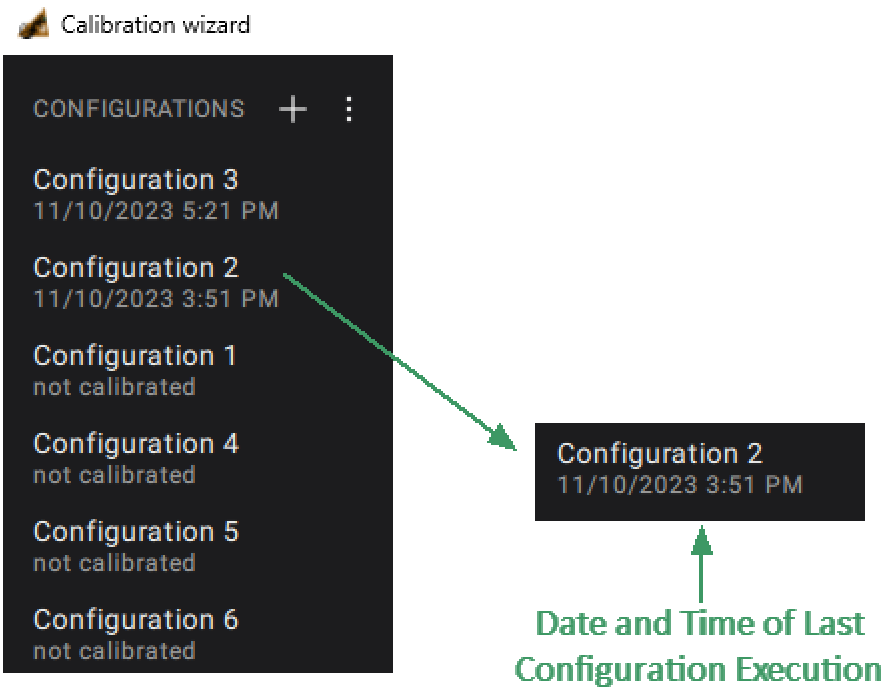


Configuration List

The Configuration List contains user-created configurations and allows to add, remove, and run the desired configuration.

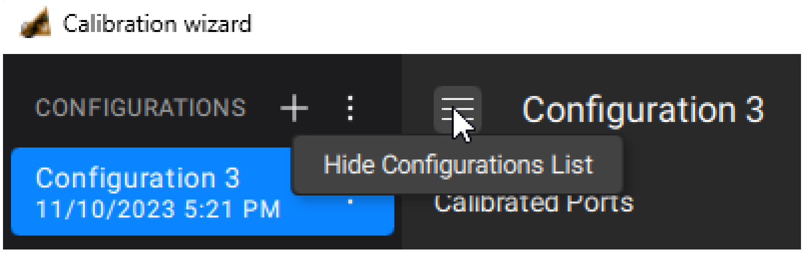
Each configuration has a unique title that can be changed if it is necessary. Information about the date and time of its last execution is displayed under the title of the configuration. If the configuration has not been performed, then **not calibrated** is displayed in the date and time line.

Configurations are sorted by date and time in descending order. If the configuration has not been not performed, it will be located at the end of the list. If there are several such configurations, they are sorted by name.



Configuration List

The configuration list can be shown/hidden by clicking on the icon in the upper left corner of the of configuration area. When performing calibration, the configuration list is automatically hidden.

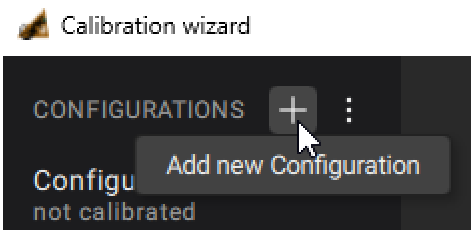


Configuration List Show/Hide Button

Adding New Configuration

1. Click on the icon in the configuration list.

   NOTE If there are no configurations in the wizard, you can create the first configuration by pressing the **ADD CONFIGURATION** button in the configuration area.



SCPI	SENSe:CORRection:CONFiguration:CREate

NOTE	Each configuration has a unique title name. You can rename a configuration title name if necessary (see Renaming Configuration).

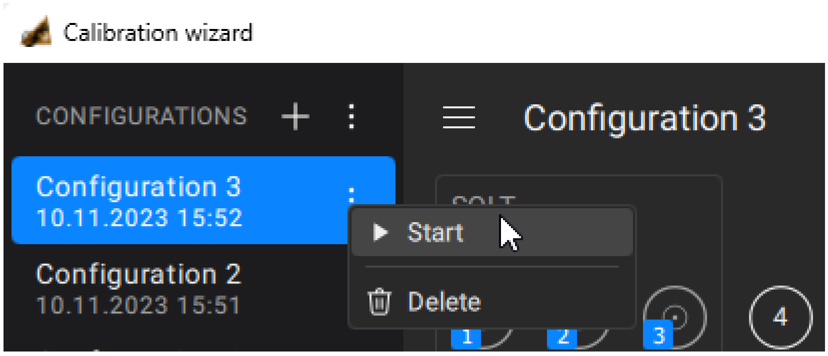
Starting Configuration from Configuration List

You can run customized configuration. To learn more about configuring, see Configuration Area.

---

1. Highlight the required configuration in the configuration list, then click the vertical-ellipsis button and click on **Start** in the pop-up window.

   NOTE If the configuration is not customized (parameters of the calibration plane for all created port groups are not set), **Start** in the pop-up window will not be available.



NOTE	You can run configuration calibration from the configuration area (see Configuration Area).

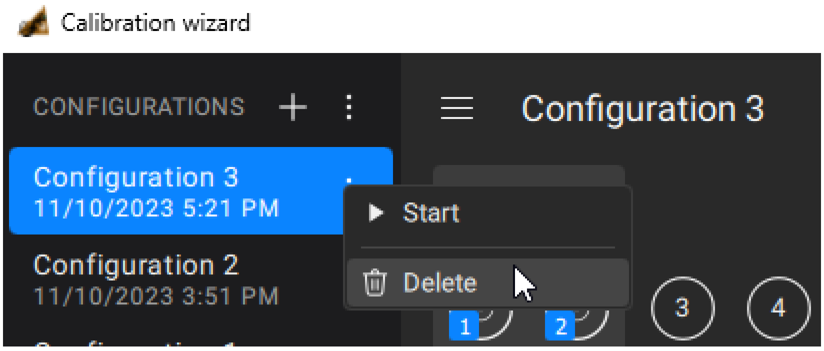
Deleting Configuration

NOTE	Deleting a configuration does not remove or turn off the correction in the channels.

1. Highlight a configuration in the wizard configuration list.

2. Click the ⋮ vertical-ellipsis button to the right of the configuration name.

3. Select **Delete** in the pop-up window.

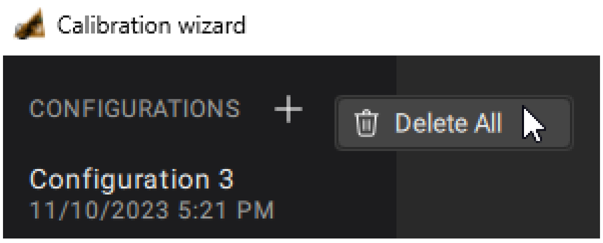


SCPI	SENSe:CORRection:CONFiguration:DELete

Deleting All Calibration Configurations

1. Click the ⋮ vertical-ellipsis button in the wizard configuration list.

2. Select **Delete All** in the pop-up window.



SCPI	SENSe:CORRection:CONFiguration:DELete:ALL

Configuration Area

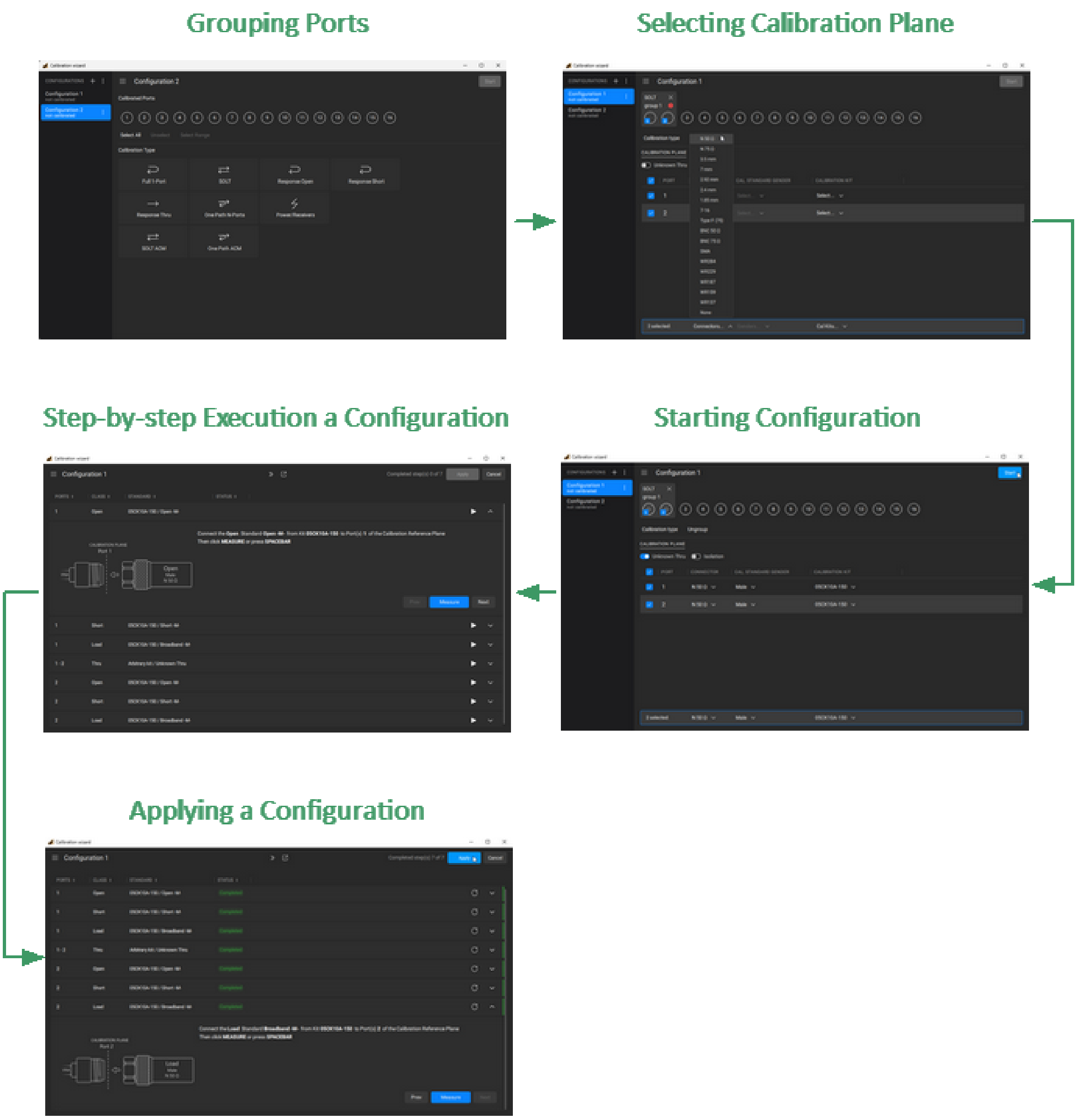
In Configuration Area the user sequentially customizes configuration and performs calibration (see figure below).The sequence of actions is the following:

1. Grouping Ports.
2. Selecting Calibration Plane.
3. Starting Configuration.
4. Step-by-Step Configuration Execution
5. Applying a Configuration.

In order to set up and execute the configuration, create a configuration (see Adding New Configuration). In Configuration Area Grouping Ports step will open.

Each step of working with the Configuration Area is described in subsections of this section below. To go to the description of the required step, you can click on the area or figure labels below.

You can also select the already created configuration (see Configuration List) and reuse it or reconfigure it (see Editing a Configuration).

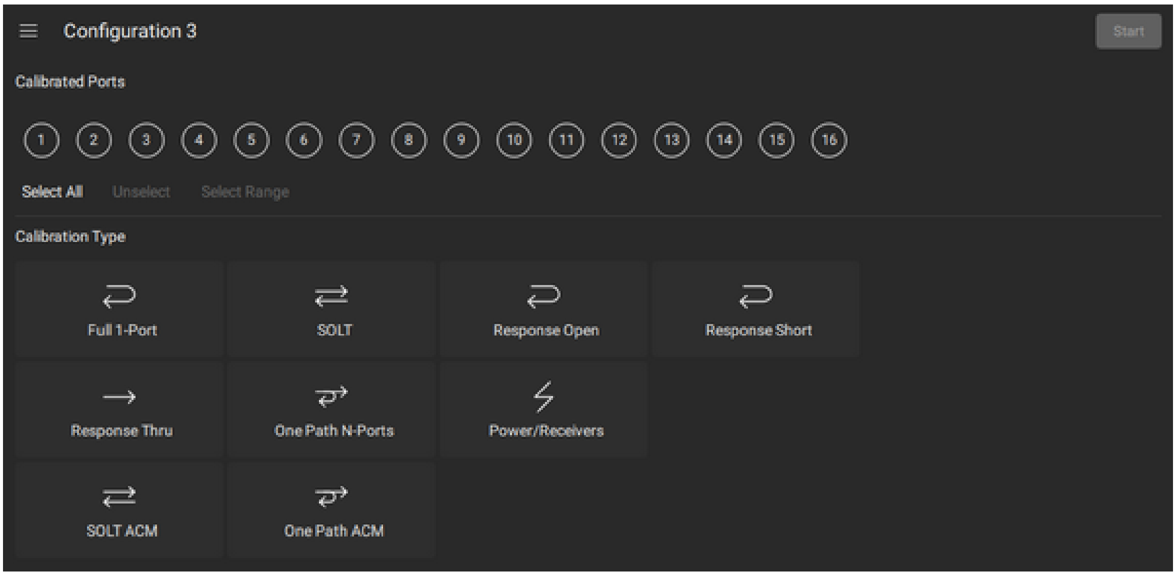


Configuration Area

Grouping Ports

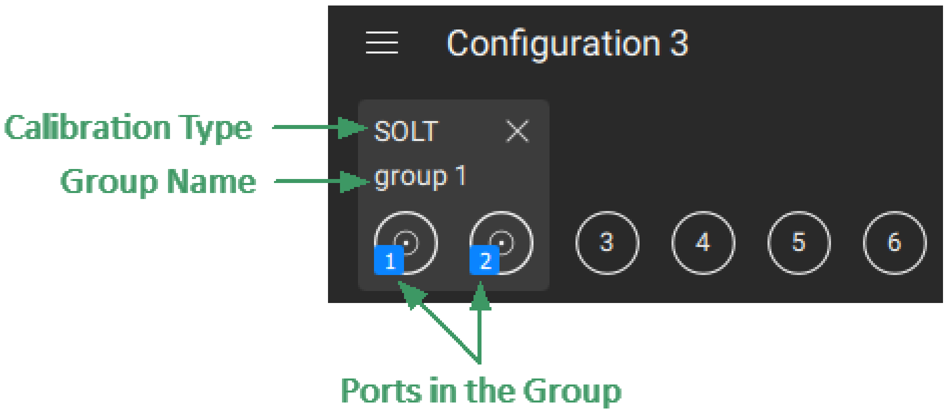
When the configuration is created, the Configuration Area will display port icons and calibration types. At this stage, it is necessary to combine the ports into a group and assign the required type of calibration to the group (see figure below). You can create several port groups with different calibration types.

NOTE	Each port can only be used in one group.



An example of combining the ports into a group (SOLT calibration is assigned for Port 1 and Port 2)

When calibration type is selected, ports are indicated grouped together in a tab labeled with group name with number and calibration type (see the example in the figure below).

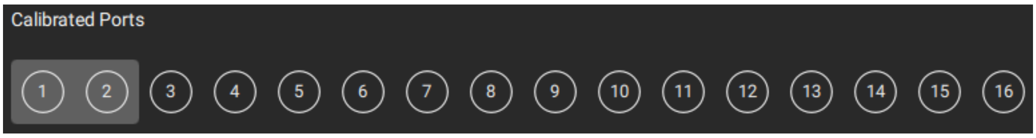


Port Group Tab

Creating a New Ports Group

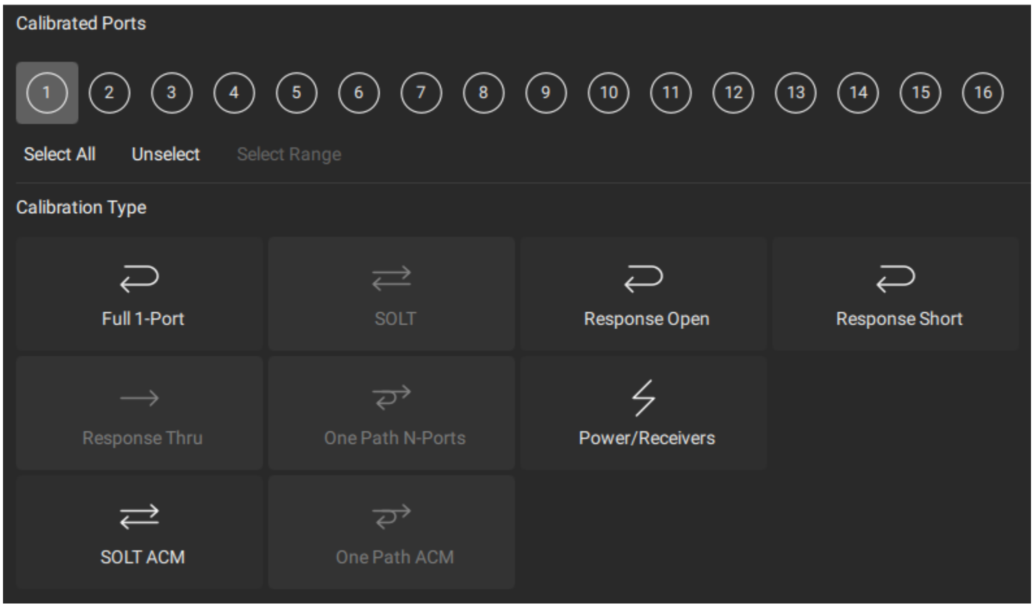
NOTE The Calibration Wizard does not allow to create S-parameter calibrations and power/receiver calibrations in the same configuration. Power/receiver calibrations should be done in a separate configuration.

1. Add the required ports to a group by clicking on the port number icons in the Calibrated Ports area.



2. Click on the button with the desired calibration type in the Calibration Type area.

NOTE If the button is not active, this type of calibration is not available for the selected number of ports.



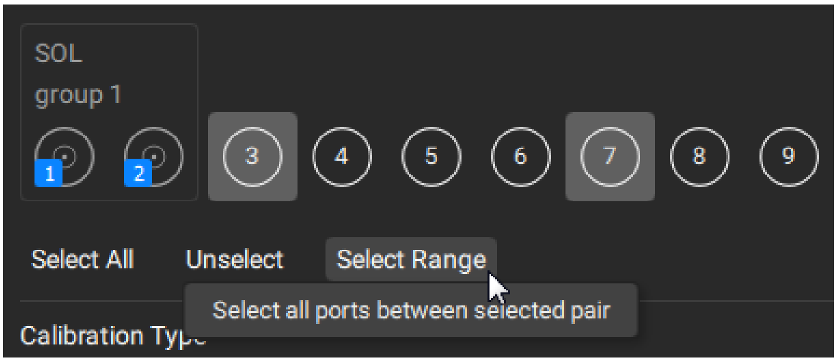
3. If you need to assign the next group of ports, highlight the required port number icons in the Calibrated Ports area and select the calibration type for them.

SCPI SENSe:CORRection:CONFiguration:GROup:CREate

NOTE

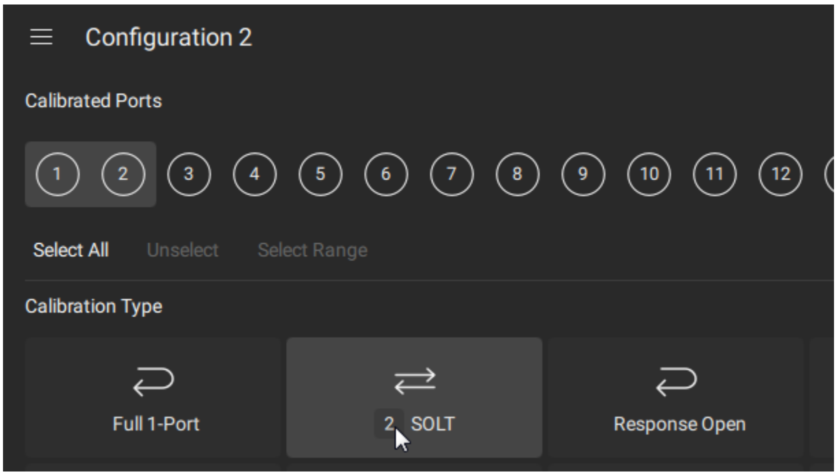
Use the buttons below to quickly select/deselect the ports in the group:

- Click on the **Select All** button to select all not occupied port.
- Click on the **Unselect** button to cancel port selection.
- Click on **Select Range** to select all ports between selected pair of ports (see figure below).

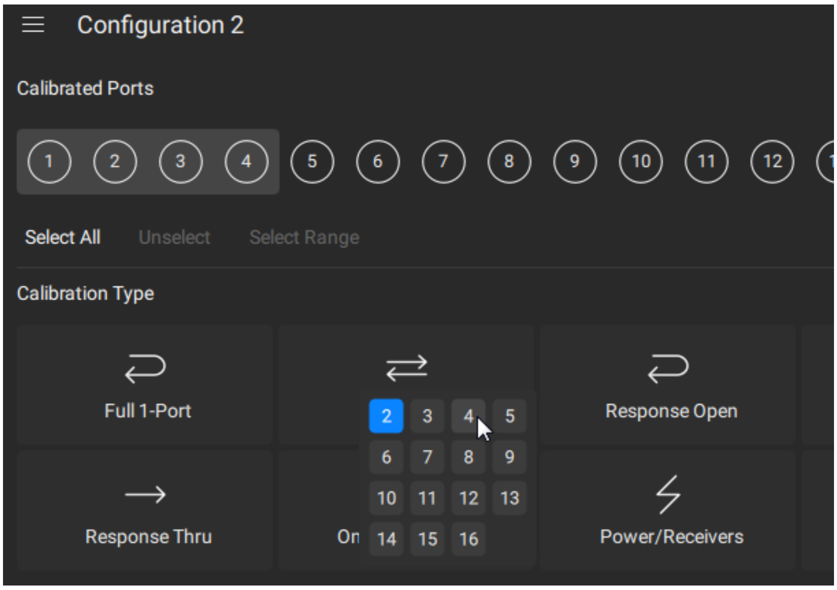


NOTE

You can assign a calibration to a group of ports by clicking on the number to the left of the calibration name in Calibration Type:

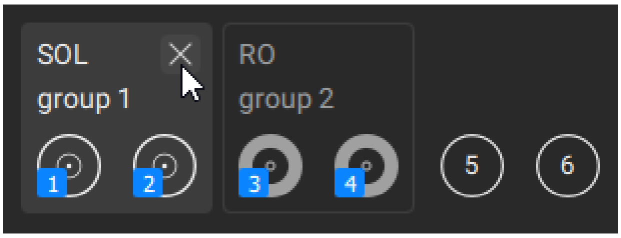


Then click on the icon with the required number of ports in the group in the pop-up window that opens (ports are combined into a group in order from the first free port).



Deleting a Ports Group

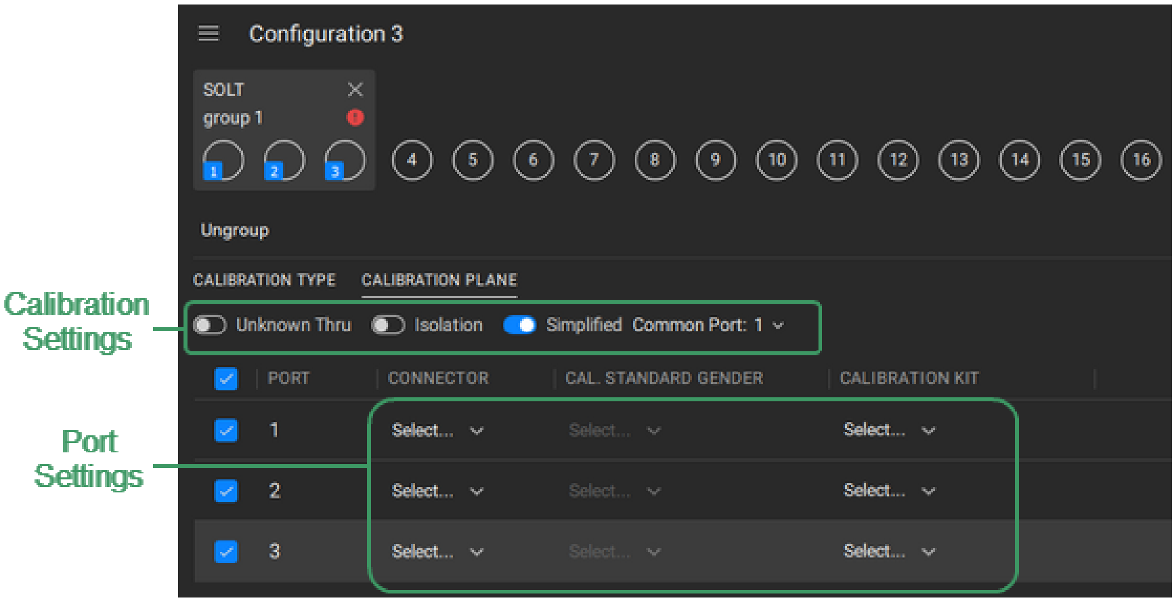
1. Click on  icon in the required Ports Group Tab in the Calibrated Ports area



SCPI	SENSe:CORRection:CONFiguration:GROup:DELete

Selecting Calibration Plane

After assigning a group, it is necessary to configure the calibration plane of the ports in the groups. The configuration is performed in the CALIBRATION PLANE tab in Configuration Area (see figure below). The plane settings to be defined depend on the selected calibration (see table below).



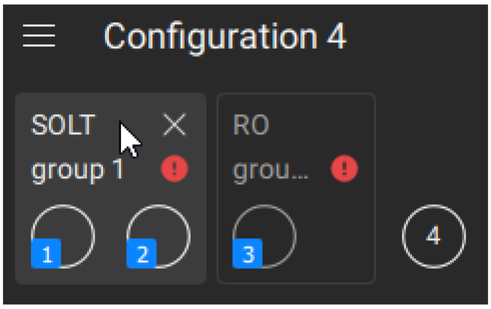
Example of Calibration Plane Definition Tab

Calibration Plane Settings

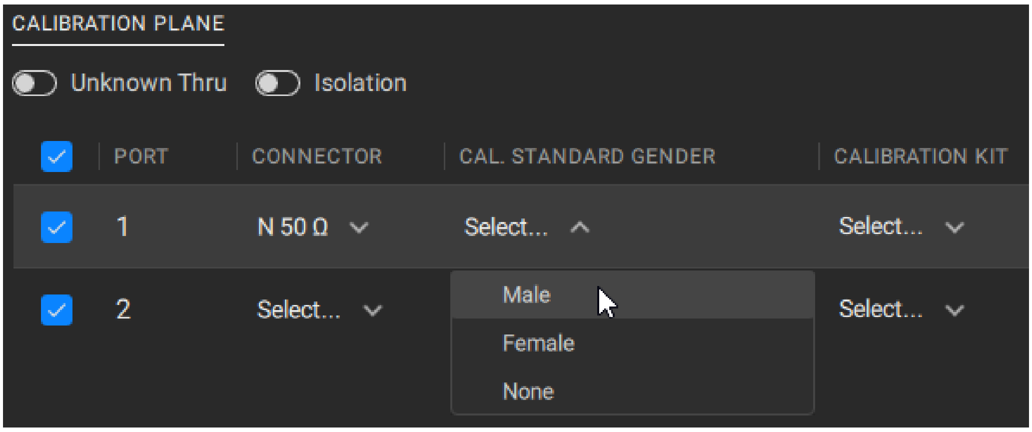
Calibration Type	Calibration Settings	Port Settings
Reflection Normalization (RO or RS)	LOAD standard	Connector type Connector gender Calibration kit
Transmission Normalization (RT)	Source Port	
Full One-Port Calibration (SOL)	—	

Selecting Calibration Plane

1 If multiple port groups are created in the Calibrated Ports area in the configuration, click on the required ports group tab (see Grouping Ports).



2 For each port in the group, click on the fields in the table and set the desired port parameters, as well as the calibration parameters in the CALIBRATION PLANE tab. For example, for a full two-port calibration (SOLT) using the calibration kit, select the desired connector type and gender, as well as the calibration kit, and if necessary, enable an unknown thru and isolation. For power calibration, select a power meter and set the loss parameters, if necessary. For a detailed description of the parameters required to set the calibration plane, refer to the description of each calibration type in Calibration Wizard.



3 If several port groups are created, then switching between groups, configure the calibration planes in each group.

NOTE If the calibration plane of at least one port is not configured, then the configuration cannot be run.

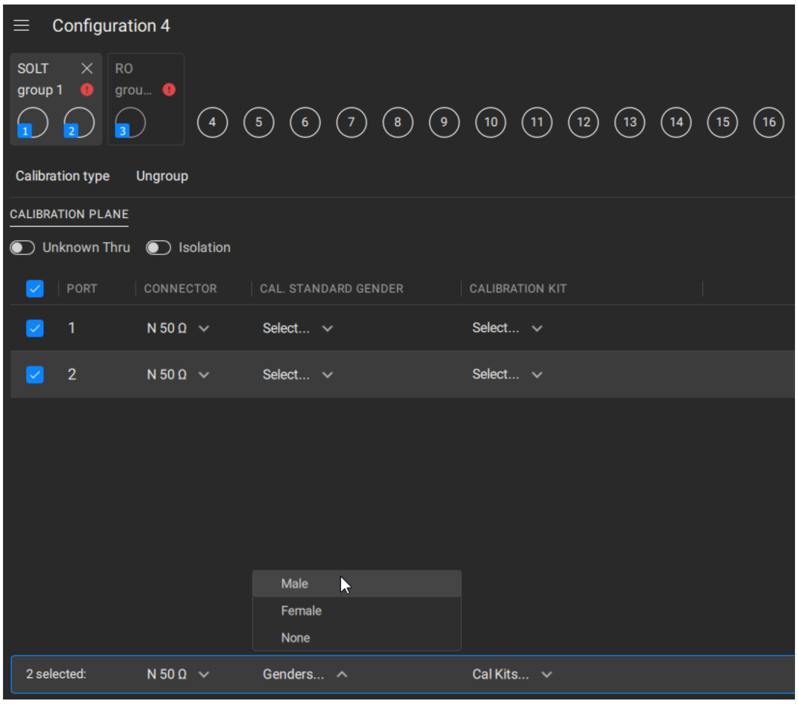
SCPI SENSe:CORRection:CONFiguration:CONNection

SENSe:CORRection:CONFiguration:GROup:CKIT:SELect

NOTE

You can install parameter to several ports at the same time:

- Select the checkboxes in the corresponding rows in a table. A line will then appear at the bottom of the table to change the parameter group.
- Click on the parameter and select the desired value from the drop-down list.



NOTE

The wizard automatically checks if the selected port connector type and gender match the calibration kit standard, or if no calibration type is selected in Power/Receiver calibration. If there is a discrepancy, e.g. there is no suitable standard in the calibration kit, the  icon will appear next to the group name. Hover the mouse over the icon to see where the discrepancies occur. The calibration procedure cannot begin if there are any discrepancy messages present.

A duplicate icon appears near the corresponding line in table.

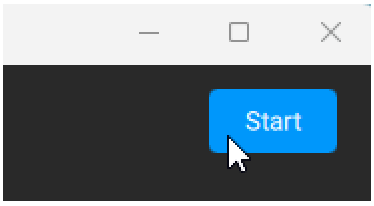
Starting Configuration

To start the calibration procedure, set the parameters of the calibration plane for all created port groups (see Selection Calibration Plane). If the calibration plane of at least one port is not configured, then the configuration cannot be run.

Starting Configuration in Configuration Area

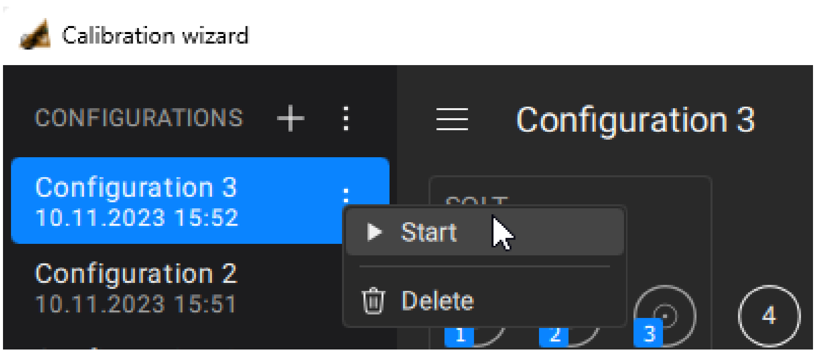
1. Click on **START** button in the Configuration area.

   NOTE The **Start** button is unavailable if there are any mistakes in the configuration of at least one of the groups (not all parameters are set, an inappropriate kit is selected, and so on).



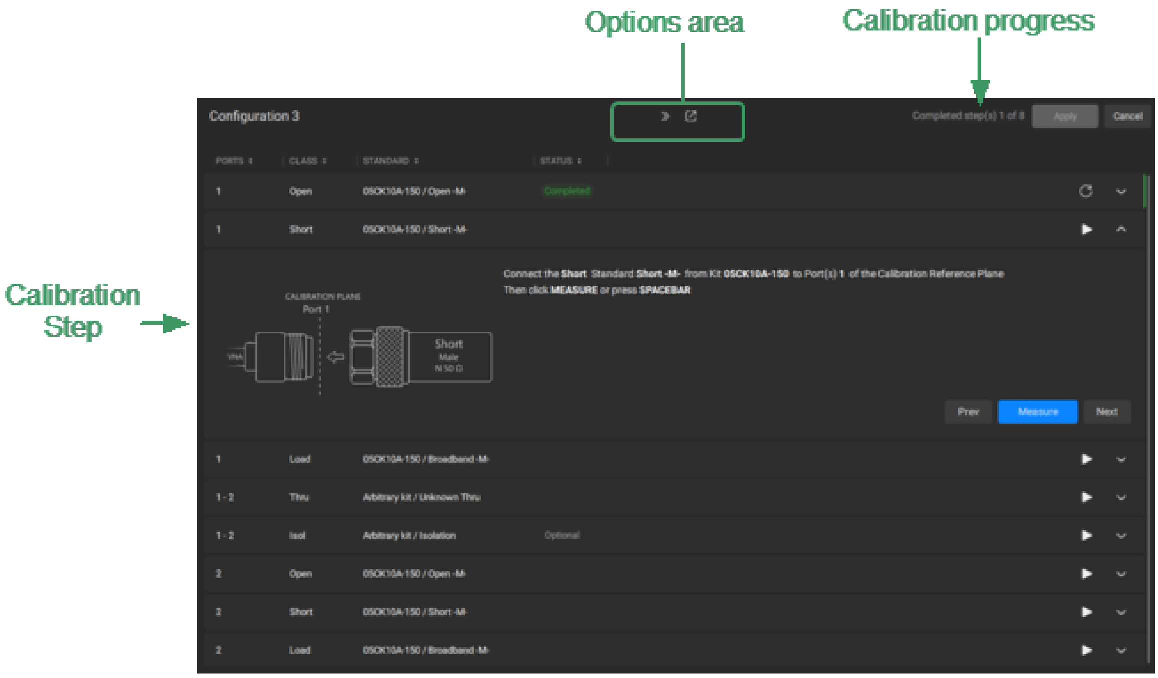
SCPI SENSe:CORRection:COLLect:STARt

NOTE You can run configuration calibration from the configuration list. Highlight the required configuration in the configuration list, then click the vertical-ellipsis button and click on **Start** in the pop-up window:



Step-by-Step Configuration Execution

After running the configuration, the Configuration Area will display calibration progress, calibration steps, options, and buttons to apply or stop the calibration (see figure below). The configuration list is hidden when starting the calibration procedure.



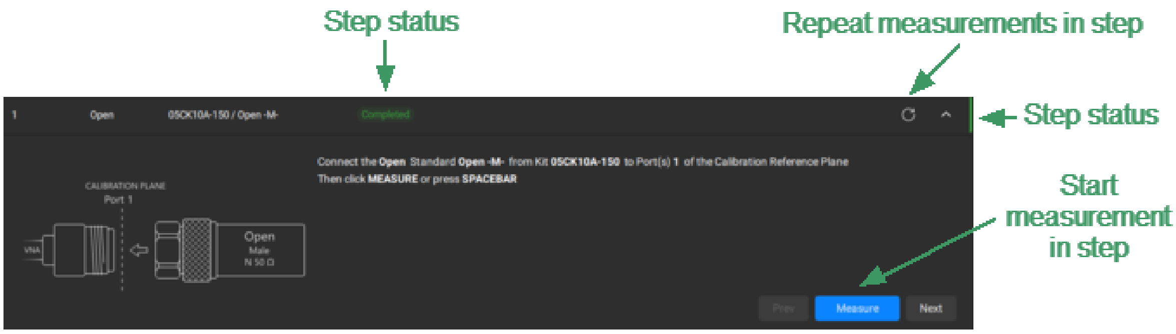
Configuration Area when Executing Calibration Steps (Configuration List is Hidden)

Calibration Step

All measurements for the selected configuration are displayed in the wizard as separate calibration steps. Calibration steps can be performed in any order. Steps marked as **Optional** in the STATUS field are performed at the user's discretion.

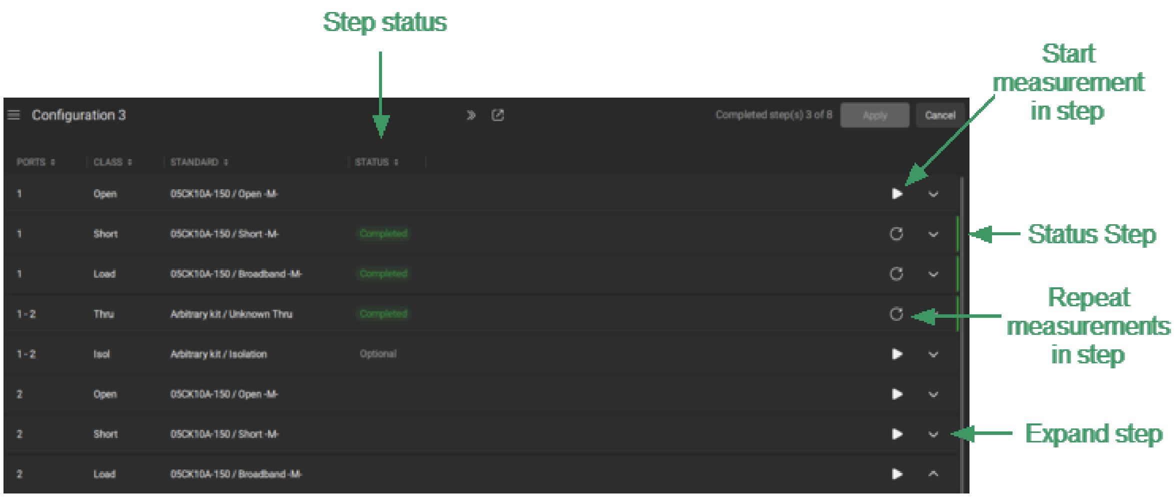
When the step is selected, traces corresponding to the step will be set in the active channel, allowing you to control the characteristics of the connected standard.

Each calibration step contains the numbers of the ports to be calibrated, necessary information about the connection of the standard(s), ACM, power meter, etc., and buttons for measuring and switching between steps (see figure below).



Expanded Calibration Step

Only one calibration step can be expanded at a time. You can perform calibration in collapsed mode when all steps are collapsed (see figure below).



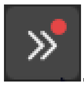
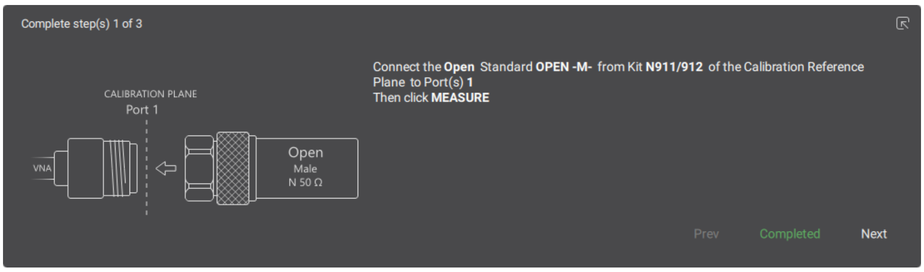
Collapsed Calibration Steps

To perform the calibration step, connect the standard to the required port(s) and perform the measurement. After performing the measurement, the step status will show **Complete** (see figures below). If necessary, the measurement of the step can be repeated.

A detailed description of each calibration type, as well as step-by-step execution of the calibration procedure, see in Calibrations Step-by-step.

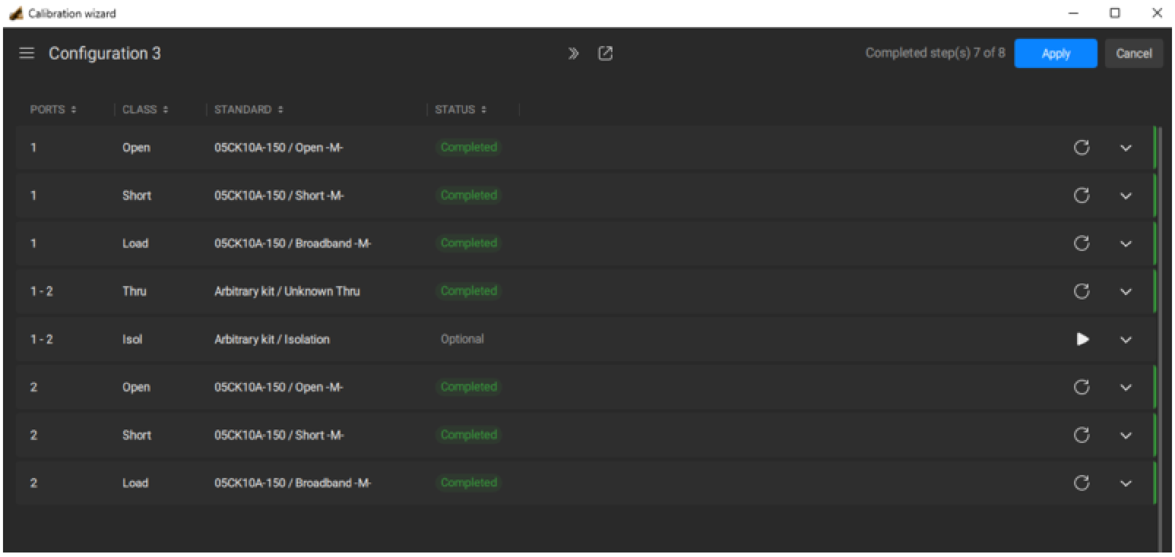
Options Area

The wizard has several calibration options. See the table below for a description of these options.

Option	Definition
AutoNext 	Automatic transition to the next calibration step after measurement. By default, the AutoNext option is set to none. After the measurement is complete, the next step automatically expands. NOTE If this option is enabled, the icon changes to .
Compact mode 	Switches the wizard window to a compact view (see figure below). In this view, the wizard contains only a line with information about the number of remaining calibration steps and the current calibration step with buttons to perform the calibration. Use this mode if during the calibration process it is necessary to monitor the channel measurement results in the main program window.  To return to the normal view, click the  icon in the upper right corner of the window.

Applying a Configuration

The configuration can be applied if all mandatory calibration steps have been completed.



Applying a Configuration (Optional Step Isolation is not performed)

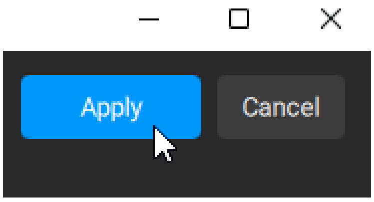
You can abort the calibration at any time. After applying calibration, the Calibration Wizard automatically closes and the correction is applied to the active channel.

NOTE	If the calibration is canceled, all standard measurement results will be lost.

Applying Configuration in Configuration Area

1 Click on **Apply** button in the Configuration area.

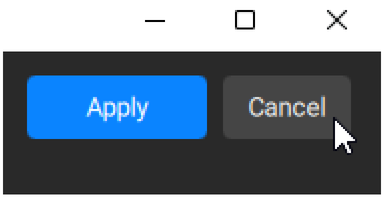
NOTE The **Apply** button is unavailable if all the mandatory calibration steps have been completed.



SCPI SENSe:CORRection:COLLect:SAVE

Aborting Configuration

1 Click on **Cancel** button in the upper right corner of the wizard.



SCPI SENSe:CORRection:COLLect:STOP

Editing a Configuration

The wizard allows to reuse the created configuration. In this case, all settings of groups and calibration planes of the specified ports will be saved.

If necessary, changes to the configuration can be made:

- Change the calibration type of a group.
- Change configuration title.
- Delete a port group (see Deleting a Ports Group).
- Add new port group (see Creating a New Ports Group).
- Change the settings of the calibration plane (see Selecting Calibration Plane).

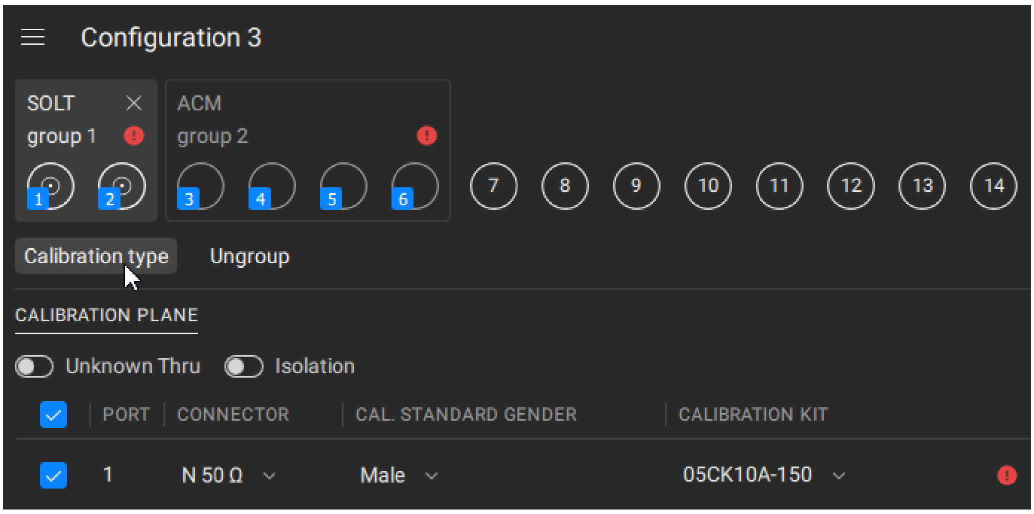
Further the calibration is performed as described in Starting Configuration.

Changing Calibration Type of a Group

---

1. Select the required group in the Calibrated Ports area.

2. Click on the **Calibration type** button as shown in figure below.



3. Select the desired calibration type from those available.

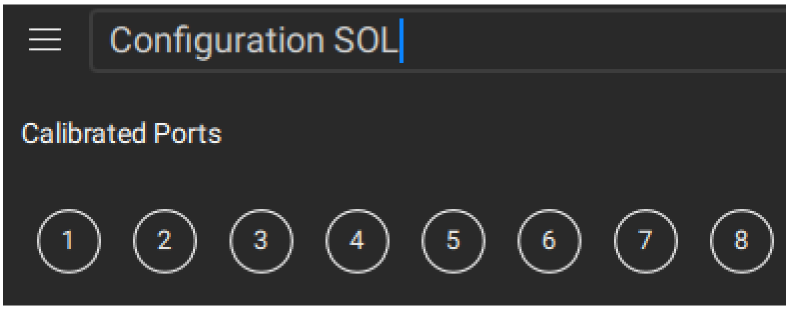
4 After making the changes, make sure make sure that all the calibration plane parameters are set (see Selecting Calibration Plane).

The wizard automatically checks if the selected port connector type and gender match the calibration kit standard. If there is a discrepancy, the  icon will appear next to the group name. Hover the mouse over the icon to see where the discrepancies occur. The calibration procedure cannot begin if there are any discrepancy messages present.

Renaming Configuration

NOTE	Configuration header name must be unique. If when renaming, the configuration name is the same as an existing one, characters **(n)** will be added to the name, where n is the number in order.

1 Click the title of the configuration title, then edit name.



Calibration Methods and Procedures

The Analyzer supports several methods of one-port and two-port calibration. The calibration methods vary by quantity and type of the standards being used, by type of error correction, and accuracy. The calibration kit is selected according to the selected calibration method. It is possible to select a calibration method from the available calibration kits. The calibration kit is selected before starting calibration (See Calibration Kit Selection). The table below presents an overview of calibration methods.

Calibration Method	Parameters	Standards	Errors	Accuracy
Reflection Normalization	S11 or S22	• SHORT or OPEN • LOAD (if optional directivity is performed)	Er1, Ed1[1] or Er2, Ed2[1]	Low
Transmission Normalization	S21 or S12	• THRU • 2 LOADs (if optional isolation calibration is performed)	Et1, Ex1[2] or Et2, Ex2[2]	Low
Full One-Port Calibration	S11 or S22	• SHORT • OPEN • LOAD	Er1, Ed1, Es1 or Er2, Ed2, Es1	High
One-Path Two-Port Calibration	S11, S21 or S12, S22	• SHORT • OPEN • LOAD • THRU	Er1, Ed1, Es1, Et1, Ex1[2] or	Medium

Calibration Method	Parameters	Standards	Errors	Accuracy
		• 2 LOADs (if optional isolation calibration is performed)	Er2, Ed2, Es2, Et2, Ex2[2]	
Two-Port TRL Calibration	S11, S21 S12, S22	• THRU or LINE • REFLECT • LINE or 2 LOADs	Er1, Ed1, Es1, Et1, El1 Er2, Ed2, Es2, Et2, El2	Very High

1. If optional directivity calibration is performed.

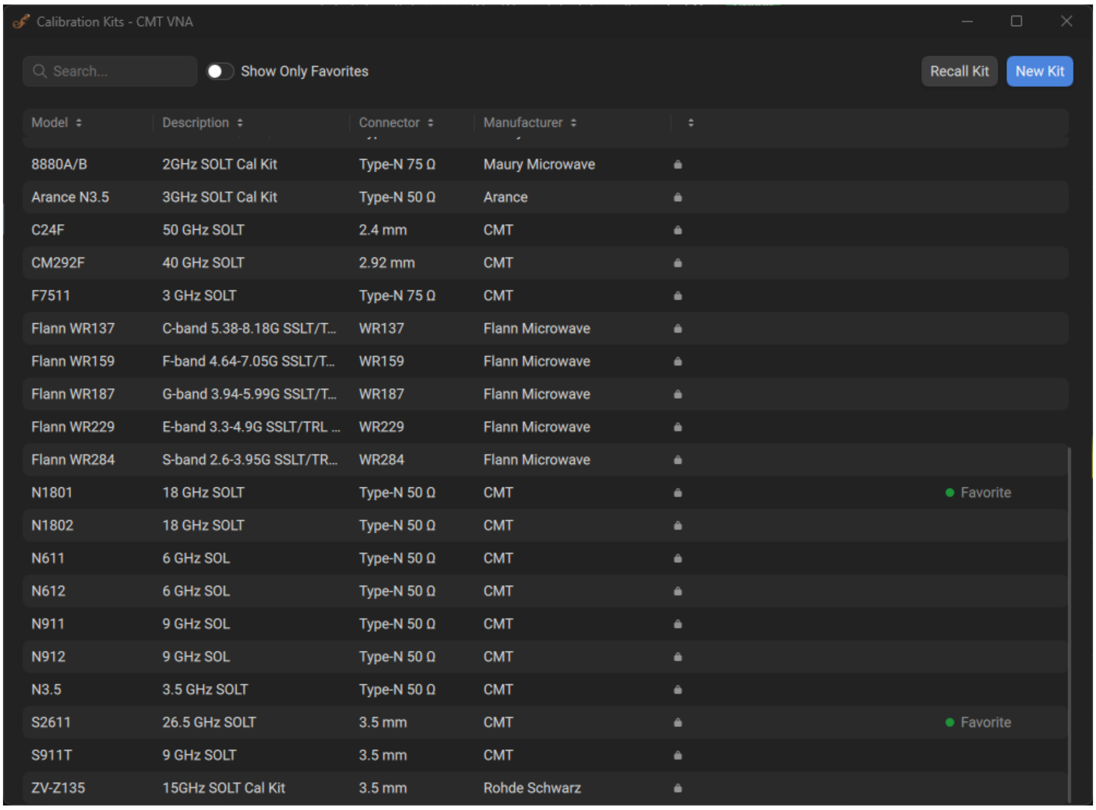
2. If optional isolation calibration is performed.

NOTE

In the waveguide path, the SOL (Short-Open-Load) calibration method can be replaced by the SSL (Short-Short-Load) and SSS (Short-Short-Short) methods. In these methods, the OPEN and LOAD standards are replaced by SHORT standards with different offsets by assigning them the corresponding classes. Both of these methods are also applicable to the coaxial path.

Calibration Kit Selection

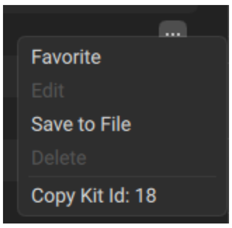
The calibration kit employed during a calibration should be selected according to the following procedure. If it is not specified in the list of the predefined calibration kits, it should be added. The procedure of adding and editing of the calibration kits is described in [Calibration Kit Management](#).



List of calibration kits

To open the list of the calibration kits:

Calibration > Cal Kit [Edit Kits...]



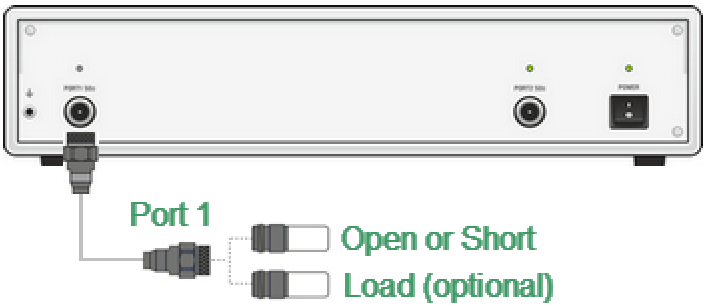
Use the ... that appear on the far right side of the window when hovering to:

- Add to favorites
- Edit
- Save to file
- Delete
- Copy Kit Id

NOTE Make sure that the selected calibration kit is marked as Favorite.

Reflection Normalization

Reflection normalization is the simplest calibration method used for reflection coefficient measurements (S11). Measurement of one standard (SHORT or OPEN) is sufficient to perform this type of calibration (See figure below).



Reflection normalization

This method is called normalization because the measured S-parameter at each frequency point is divided (normalized) by the corresponding S-parameter of the calibration standard. Reflection normalization corrects the reflection tracking error (**Er**) only. This constrains the accuracy of the method.

Normalization eliminates frequency-dependent attenuation and phase offset in the measurement circuit but does not compensate errors of directivity and mismatch.

NOTE	Reflection normalization can also be referred to as **response open** or **response short** calibration depending on the standard being used: OPEN or SHORT.

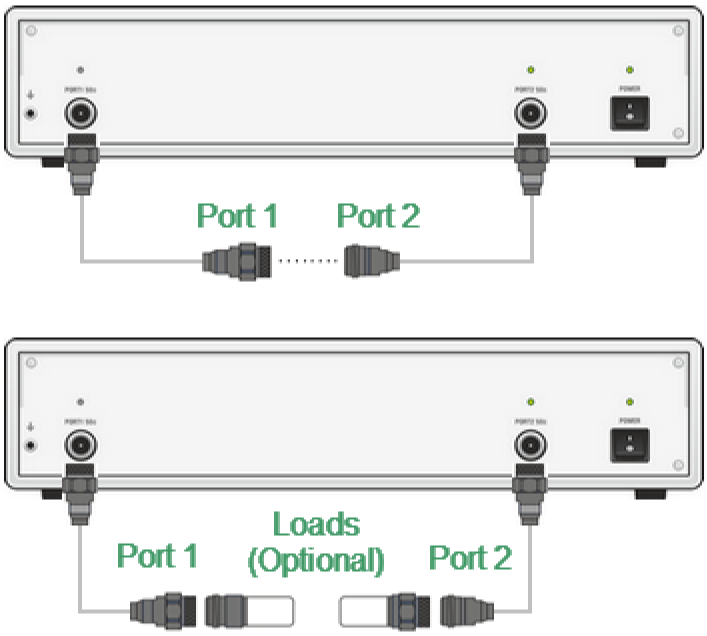
Transmission Normalization

Transmission normalization is the simplest calibration method used for transmission coefficient measurements (S21 or S12). Measurement of one THRU standard is enough to perform this type of calibration (See figure below). This method is called normalization because the measured S-parameter at each frequency point is divided (normalized) by the corresponding S-parameter of the calibration standard. Transmission normalization corrects the transmission tracking error (**Et)** only. This constrains the accuracy of the method.

NOTE	Transmission normalization can also be referred to as **response thru** calibration.

An optional isolation calibration can be performed by measurement of two LOAD standards connected to both test ports of the analyzer. In this case, the isolation error (**Ex**) is additionally corrected in the transmission normalization.

NOTE	For isolation calibration, set a narrow IF bandwidth and firmly attach the cables.

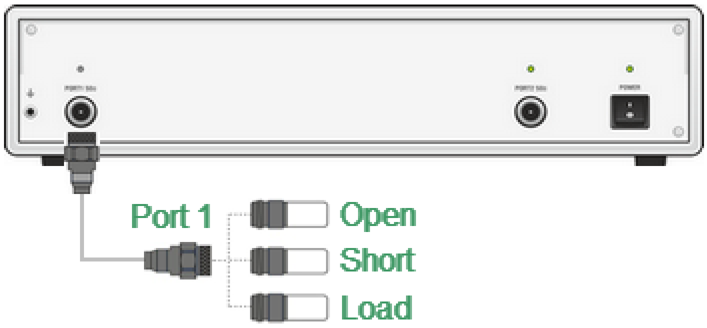


Transmission normalization

Before starting calibration perform, select an active channel, set the parameters of the channel (frequency range, IF bandwidth, etc), and select the calibration kit.

Full One-Port Calibration

Full one-port calibration (SOL) is used for reflection coefficient measurements (S11 or S22). The three calibration standards (SHORT, OPEN, LOAD) are measured (See figure below) in the process of this calibration. Measurement of the three standards allows for acquisition of all the three error terms (**Ed**, **Es**, and **Er**) of a one-port model. Full one-port calibration is a highly accurate method for one-port reflection measurements.



One-Path Two-Port Calibration

A one-path two-port calibration combines full one-port calibration with transmission normalization. This method allows for a more accurate estimation of transmission tracking error (**Et**) than using transmission normalization.

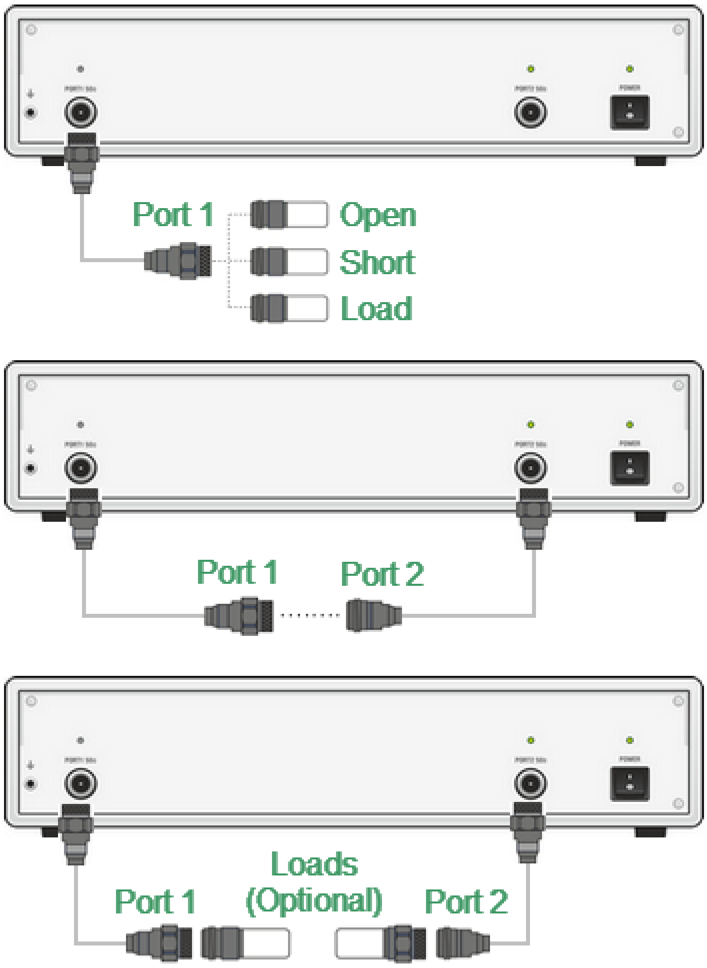
One-path two-port calibration involves connection of the three standards to the source port of the Analyzer (as for one-port calibration) and a THRU standard connection between the calibrated source port and the other receiver port (See figure below).

One-path two-port calibration allows for correction of **Ed**, **Es**, and **Er** error terms of the source port and a transmission tracking error term (**Et**). This method does not derive source match error term (**El**) of a two-port error model.

An optional isolation calibration can be performed by measurement of two LOAD standards connected to both test ports of the Analyzer. In this case, the isolation error (**Ex**) is additionally corrected in the one-path two-port calibration.

NOTE	For isolation calibration, set a narrow IF bandwidth and firmly attach the cables.

One-path two-port calibration is used for measurements of the parameters of a non-reciprocal DUT such as amplifiers in one direction, e.g. S11 and S21.



One-path two-port calibration

Before starting calibration perform, select an active channel, set the parameters of the channel (frequency range, IF bandwidth, etc), and select the calibration kit.

Waveguide Calibration

The Analyzer supports the following calibration methods in a waveguide environment:

- Reflection Normalization or Transmission Normalization
- Full One-Port Calibration
- One-Path Two-Port Calibration

Error Correction Status

The error correction status is indicated for each trace individually. There is also a general status of error correction for all traces of a channel.

General error correction status

The general error correction status for all S-parameter traces of a channel is indicated in the specific field on a channel status bar (See General error correction status table). For the channel status bar description, see Channel Status Bar.

Symbol	Definition	Note
Cor	Error correction is enabled. The stimulus settings are the same for the measurement and the calibration.	If the function is active for all traces — black characters on a gray background. If the function is active only for some of the traces (other traces are not calibrated) — white characters on a red background.
C?	Error correction is enabled. The stimulus settings are not the same for the measurement and the calibration. Interpolation is applied.	
C!	Error correction is enabled. The stimulus settings are not the same for the measurement and the calibration. Extrapolation is applied.	
Off	Error correction is turned off.	For all traces. White characters on a red background.
---	No calibration data. No calibration was performed.	

Trace error correction status

The error correction status for each individual trace is indicated in the trace status field (See table below).

Symbols	Definition
RO	OPEN response calibration
RS	SHORT response calibration
RT	THRU response calibration
OP	One-path two-port calibration
F1	Full one-port (SOL) calibration
F2	Full two-port (SOLT) or TRL calibration
SMC	Scalar mixer calibration

Error Correction Disabling

This feature allows to disable the error correction function, which automatically becomes enabled after completion of calibration by any method.

To disable and enable the error correction function:

Calibration > Settings > Correction

SENS:CORR:STAT	Turns the S-parameter error correction ON/OFF.

Calibration Trigger Source

The function sets the trigger source to start measuring the calibration standards. If an Internal source is selected, the calibration starts immediately. If the source is **System**, the system trigger is used to start the calibration. The source of the system trigger is set by the softkey:

Trigger > Trigger Source > [Internal | External | Manual | Bus]

If a system trigger is used, the averaging trigger function and the external trigger event function set On Point affect the start of the calibration in the same way as during standard measurements. When using a system trigger, the trigger source Bus should not be used, otherwise the software may be blocked.

NOTE	This function does not apply to calibration using the ACM, power calibration, receiver calibration. In these calibrations, the internal trigger is always used.

Measurement Data Analysis

The following section describes the process of Measurement Data Analysis using the Analyzer.

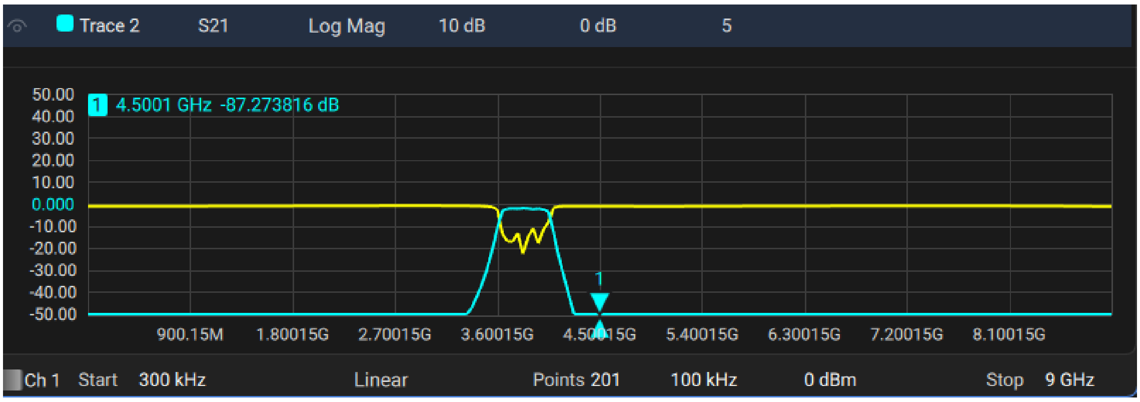
Special software marker tools are used to read and look up the numerical values of the stimulus and the measured value on selected points on the graph. For a detailed description, see Markers.

This section also contains information about the various functions and tools used to analyze measurements.

- Memory Trace Function is used to save data traces and perform mathematical operations between memory and data traces.
- Trace Hold is used to hold the maximum or minimum values of the trace.
- Fixture Simulation is used to simulate measurement conditions that differ from real measurement conditions.
- Time Domain Transformation is used to convert the measured characteristics in the frequency domain into the circuit response in the time domain.
- Time Domain Gating (except M models) is used to eliminate unwanted responses in the time domain.
-
- A function of pass/fail determination for the trace of the measurement data according to various criteria:
  1. Limit Test is used to compare the trace of the measured value with the limit line.
  2. Ripple Limit Test is used to check the value of the ripple trace with user-defined ripple limits
  3. Peak Limit Test is used to check if the peak of the trace of the set polarity falls within the limits for the peak.

Markers

A marker is a tool for numerical readout of a stimulus value and value of the measured parameter in a specific point on the trace. Up to 16 markers can be activated on each trace. A trace with a marker is shown in the figure below.



Trace with marker

The markers allow to perform the following tasks:

- Reading absolute values of a stimulus and a measured parameter in selected points on the trace.
- Reading relative values of a stimulus and a measured parameter related to the reference point.
- Search for specific points on the trace (minimum, maximum, target level, etc.).
- Determining trace parameters (statistics, bandwidth, etc.).
- Editing stimulus parameters using markers.

Markers can have the following indicators:

1 ∇	Symbol and number of the active marker on a trace.
Δ 2	Symbol and number of the inactive marker on a trace.
▲	Symbol of the active marker on a stimulus axis.
Δ	Symbol of the inactive marker on a stimulus axis.

The marker data field contains the marker number, stimulus value, and the measured parameter value. The number of the active marker is highlighted in an inverse color.

The marker data field contents vary depending on the display format (rectangular or circular):

- In rectangular format, the marker shows the measurement parameter value plotted along Y-axis in the active format (See the table below).

Format Type Description	Label	Data Type (Y-axis)	Measurement Unit (Y-axis)
Logarithmic Magnitude	**Log Mag**	S-parameter logarithmic magnitude: $20 \cdot \log\lvert S\rvert$, $\lvert S\rvert = \sqrt{a^2 + b^2}$	Decibel (dB)
Voltage Standing Wave Ratio	**SWR**	$\frac{1+\lvert S\rvert}{1-\lvert S\rvert}$	Dimensionless value
Phase	**Phase**	S-parameter phase from –180° to +180°: $\frac{180}{\pi} \cdot arctg\frac{b}{a}$	Degree (°)
Expanded Phase	**Expand Phase**	S-parameter phase, measurement range expanded to from below –180° to over +180°	Degree (°)
Group Delay	**Group Delay**	Signal propagation delay within the DUT: $-\frac{d\varphi}{d\omega}$, $\varphi = arctg\frac{b}{a}$, $\omega = 2\pi \cdot f$	Second (sec.)
Linear Magnitude	**Lin Mag**	S-parameter linear magnitude: $\sqrt{a^2 + b^2}$	Dimensionless value
Real Part	**Real**	S-parameter real part: $a = re(S)$	Dimensionless value
Imaginary Part	**Imag**	S-parameter imaginary part: $b = im(S)$	Dimensionless value

- In circular format, the marker shows two or three values listed in the table below.

Label	Marker Readings (Measurement Unit)		
	Reading 1	**Reading 2**	**Reading 3**
Smith (Lin)	Linear magnitude	Phase (°)	—
Smith (Log)	Logarithmic magnitude (dB)	Phase (°)	—
Smith (Re/Im)	Real part	Imaginary part	—
Smith (R + jX)	Resistance (Ω)	Reactance (Ω)	Equivalent capacitance or inductance (F/H)
Smith (G + jB)	Conductance (S)	Susceptance (S)	Equivalent capacitance or inductance (F/H)
Polar (Lin)	Linear magnitude	Phase (°)	—
Polar (Log)	Logarithmic magnitude (dB)	Phase (°)	—
Polar (Re/Im)	Real part	Imaginary part	—

Marker Addition

To enable a new marker:

Marker > Add

Or use the add marker button in the quick access toolbar.

CALC:MARK	Turns the marker ON/OFF.

NOTE	The new marker appears as the active marker in the middle of the stimulus axis. The input field for the marker stimulus value is activated.

Marker Deletion

To delete a marker:

Marker > Delete

Or use the delete marker button in the quick access toolbar.

Marker Activation

To activate a marker by its number, use the following softkeys:

Marker > Active

And use the dropdown menu to select the marker.

CALC:MARK:ACT	Sets the active marker.

NOTE	A marker can be activated by clicking on it.

Marker Stimulus Value Setting

The active marker must be selected before setting the marker stimulus value. The stimulus value must be set by entering the numerical value from the keyboard, by arrows, by dragging the marker using the mouse (See Marker Stimulus Value Setting), or by enabling the search function (See Marker Position Search Functions).

To set the marker stimulus value:

Markers > Stimulus

and either type in the desired value or use the arrows in the textbox.

CALC:MARK:X	Sets or reads out the stimulus value of the marker.

Reference Marker Feature

The reference marker feature allows to view the data relative to the reference marker. Other markers readings are represented as delta relative to the reference marker. The reference marker shows the absolute data and is indicated with «R» symbol instead of a number. Enabling of a reference marker turns all the other markers to relative display mode.

Reference marker can be indicated on the trace as follows:

R ∇	Symbol of the active reference marker on a trace.
Δ R	Symbol of the inactive reference marker on a trace.

The reference marker displays the stimulus and measurement absolute values. The rest of the markers display the relative values:

- Stimulus value (ΔX in the figure above) is the difference between the absolute stimulus values of this marker and the reference marker.
- Measured value (ΔY in the figure above) is the difference between the absolute measurement values of this marker and the reference marker.

To enable/disable the reference marker, use the following softkeys:

Marker > Ref Marker [ON|OFF]

CALC:MARK	Turns the marker ON/OFF.
CALC:MARK:ACT	Sets the active marker.
CALC:MARK:REF	Turns the reference marker ON/OFF.

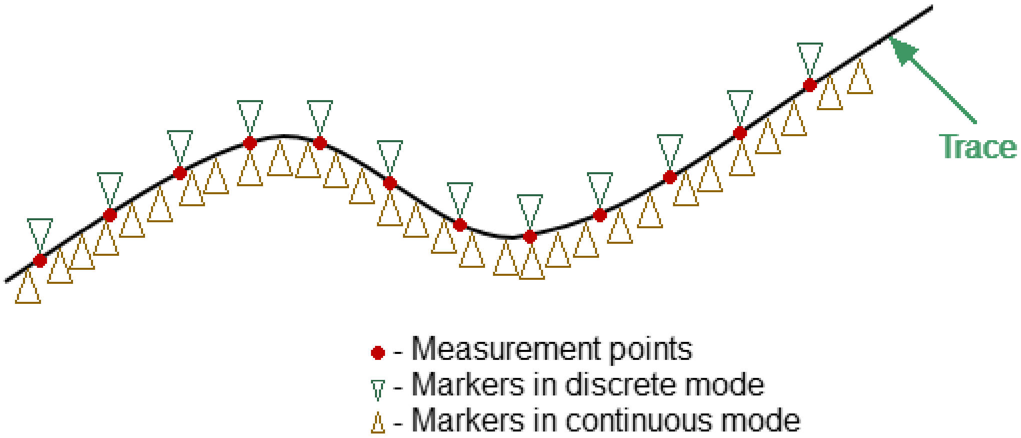
Marker Properties

The following section describes marker properties:

-
- Marker Discrete Mode is the moving of the marker only between actual measurement points.
- Memory Trace Value Display is the ability to turn on the memory trace marker values if a memory trace is available.

Marker Discrete Mode

By default, the marker can be moved along the values interpolated between measurement points. To move the marker only between actual measurement points, enable the marker discrete mode.



Marker Discrete and Continuous Modes

To enable / disable discrete mode:

Marker > Properties > Discrete [ON | OFF]

CALC:MARK:DISC	Turns the marker discrete mode ON/OFF.

Memory Trace Value Display

By default, the marker values of the data traces (not memory traces) are displayed on the screen. The display of memory trace maker values can be enabled, if a memory trace is available.

To enable/disable the display of memory trace marker values, toggle the softkey:

Marker > Properties > Memory Value [ON | OFF]

Marker Position Search Functions

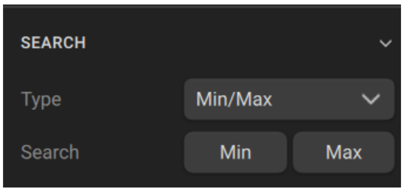
The marker position search function allows to find the following values on a trace:

- Maximum value
- Minimum value
- Peak value
- Target level

This section contains information about search tracking mode (See Search Tracking) and on the function used to set the search range of the marker position (See Search Range).

Maximum and Minimum Search Functions

Maximum and minimum search functions are used to determine the maximum and minimum values of the measured parameter and move the marker to these positions on the trace (See figure below).



To find the maximum or minimum values on a trace, use the following softkeys:

Marker Search > Type [Min/Max]

Search [Min] [Max]

CALC:MARK:FUNC:EXEC	Executes the marker search according to the specified criterion.
CALC:MARK:FUNC:TYPE	Selects the type of the marker search.

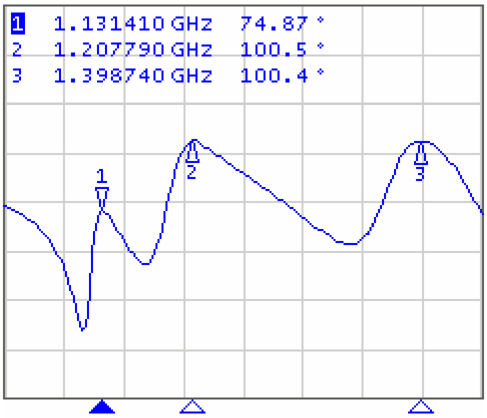
NOTE	Activate the marker before starting maximum or minimum search (See Marker Activation). In Smith chart and polar formats, the search is executed for the first marker value.

Search for Peak

Peak search function is used to determine the peak value of the measured parameter and move the marker to this position on the trace.

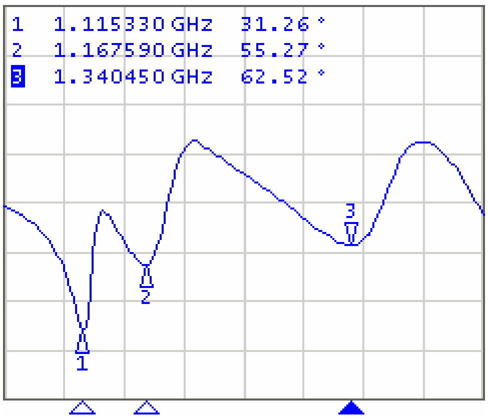
Peak is a local extreme of the trace.

Peak is considered **positive** if the value of the peak is greater than the values of the adjacent points (See figure below).



Positive peaks

Peak is considered **negative** if the value of the peak is smaller than the values of the adjacent points (See figure below).



Negative peaks

Peak excursion is the smallest of the absolute differences between the response values in the peak point and the two adjoining peaks of the opposite polarity.

The peak search is executed only for the peaks meeting the following conditions:

- The peaks must have the polarity (positive, negative, or both) specified by the user.
- The peaks must have a peak deviation no less than the value assigned by the user.

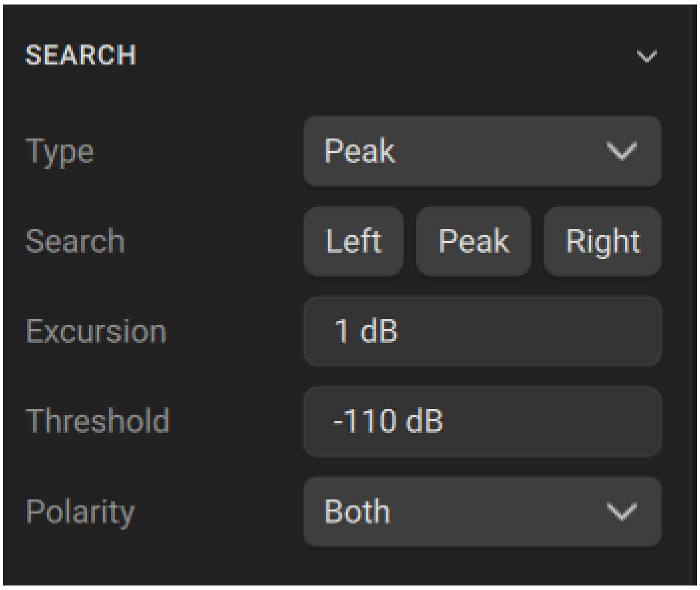
The following options for the peak search are available:

- search for nearest peak
- search for greatest peak
- search for left peak
- search for right peak

The nearest peak is a peak that is located most near to the current position of the marker along the stimulus axis.

The greatest peak is a peak with maximum or minimum value, depending on the current polarity settings of the peak.

NOTE	Finding the greatest peak is different form finding the maximum or minimum, as the peak cannot be located at the trace's limit points, even if those points have a maximum or minimum value.



Marker Search Peak Search Menu

To set the polarity of the peak:

Search > Type: Peak > Polarity [Positive|Negative|Both]

CALC:MARK:FUNC:PPOL	Selects the peak polarity when the marker peak search is performed by the CALC:MARK:FUNC:EXEC command.

To enter the peak excursion value:

Search > Type: Peak > Excursion

Then enter the value using the numerical keypad, or the «↑», «↓» arrows.

CALC:MARK:FUNC:PEXC	Sets or reads out the peak excursion value when the marker peak search is performed by the CALC:MARK:FUNC:EXEC command.

To activate the nearest peak search:

Search > Type: Peak > Search [Peak]

To activate the left peak search:

Search > Type: Peak > Search [Left]

To activate the right peak search:

Search > Type: Peak > Search [Right]

CALC:MARK:FUNC:EXEC	Executes the marker search according to the specified criterion.
CALC:MARK:FUNC:TYPE	Selects the type of the marker search, which is performed by the CALC:MARK:FUNC:EXEC command.

NOTE

Activate the marker before starting peak search (see Marker Activation).

In Smith chart and Polar formats, the search is executed for the first marker value.

Search for Target Level

The target level search function is used to locate the marker with the given level of the measured parameter.

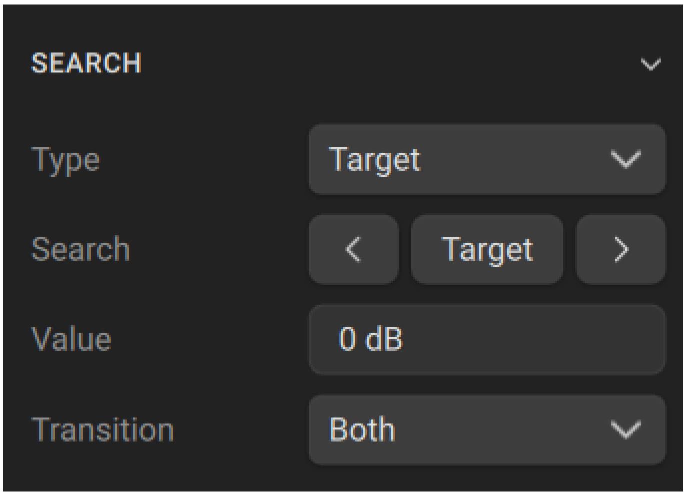
The trace can have two types of transition at the points where the target level crosses the trace:

- Transition type is positive if the function derivative (trace slope) is positive at the intersection point with the target level.
- Transition type is negative if the function derivative (trace slope) is negative at the intersection point with the target level.

Target level search is performed only for intersection points that have a user-selected specific transition polarity (positive, negative, or both).

The following options for the target level search are available:

- search for nearest target
- search for left target
- search for right target



Marker Search Target Search Menu

To set the transition polarity:

Search > Type: Target > Transition [Positive | Negative | Both]

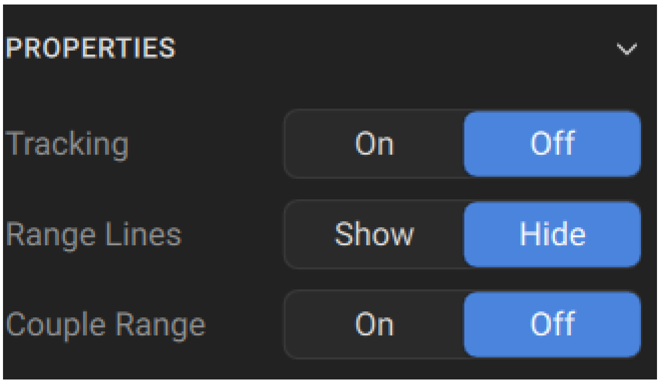
CALC:MARK:FUNC:TTR	Selects the type of the target transition when the marker transition search is performed by the CALC:MARK:FUNC:EXEC command.

	To enter the target level value: **Search > Type: Target > Value** Then enter the value using the numerical keypad, or the «↑», «↓» arrows.
CALC:MARK:FUNC:TARG	Sets or reads out the target value when the marker target search is performed by the CALC:MARK:FUNC:EXEC command.
	To activate the nearest target search: **Search > Type: Target > Search [Target]**
	To activate the left target search: **Search > Type: Target > Search [Left]**
	To activate the right target search: **Search > Type: Target > Search [Right]**
CALC:MARK:FUNC:EXEC	Executes the marker search according to the specified criterion.
CALC:MARK:FUNC:TYPE	Selects the type of the marker search, which is performed by the CALC:MARK:FUNC:EXEC command.

NOTE	Activate the marker before starting target level search (see Marker Activation). In Smith chart and Polar formats, the search is executed for the first marker value.

Search Tracking

The marker position search function, by default, can be initiated by any press of the search key. Search tracking mode performs continuous marker position search, until this mode is disabled.



Marker Search Properties

To enable/disable search tracking mode:

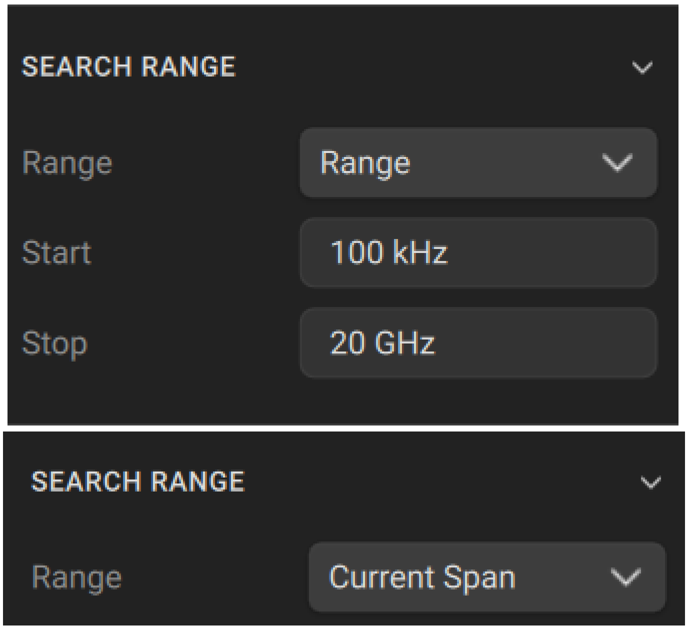
Search > Properties > Tracking [On|Off]

CALC:MARK:FUNC:TRAC	Sets or reads out the target value when the marker target search is performed by the CALC:MARK:FUNC:EXEC command.

Search Range

The search range for the marker position search can be set by setting the stimulus limits. This function includes the following additional features:

- Search range coupling, which allows to define the same search range for all the traces of a channel.
- Vertical line indication of the search range limits.



Marker Search Range

To enable/disable the search range:

Search > Search Range > Current Span | Range

CALC:MARK:FUNC:DOM	Turns the state of the arbitrary range when executing the marker search ON/OFF.

To set the search range limits, use the following softkeys:

Search > Search Range > Start
Search > Search Range > Stop

CALC:MARK:FUNC:DOM:STAR	Sets or reads out the start value of the marker search range.
CALC:MARK:FUNC:DOM:STOP	Sets or reads out the stop value of the marker search range.

Marker Math Functions

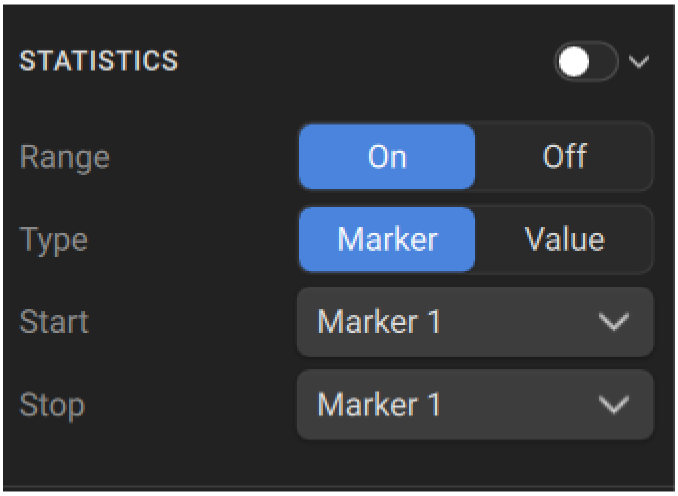
Marker math functions use markers to calculate various trace characteristics. Four marker math functions are available:

- Statistics
- Bandwidth Search
- Flatness
- RF Filter

Trace Statistics

The trace statistics feature allows to determine and view trace parameters, such as mean, standard deviation, and peak-to-peak.

The range of trace statistics can be defined by two markers.



Math Statistics When On

Trace Statistics parameter

Symbol	Definition	Formula
mean	Arithmetic mean	$M = \frac{1}{N} \cdot \sum_{i=1}^{N} x_i$
s.dev	Standard deviation	$\sqrt{\frac{1}{N-1} \cdot \sum_{i=1}^{N} (x_i - M)^2}$
p-p	Peak-to-Peak: difference between the maximum and minimum values	Max – Min

To enable/disable trace statistics function:

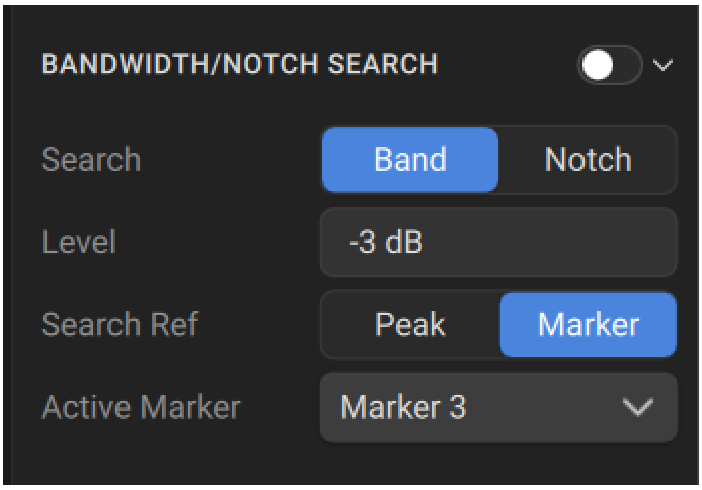
Math > Statistics

	And use the toggle to turn it ON/OFF.
CALC:MST	Turns the math statistics display ON/OFF.
	To enable/disable trace statistics range: **Math > Statistics > Range [ON|OFF]**
CALC:MST:DOM	Selects either the partial frequency range or the entire frequency range to be used for math statistic calculation.
	To set the start/stop markers of the statistics range: **Math > Statistics > Start** **Math > Statistics > Stop**
CALC:MST:DOM:STAR	Sets or reads out the number of the marker, which specifies the start frequency of the math statistics range.
CALC:MST:DOM:STOP	Sets or reads out the number of the marker, which specifies the stop frequency of the math statistics range.

Bandwidth Search

The bandwidth search function allows to determine and view the following parameters of a passband or a stopband: bandwidth, center frequency, lower frequency, higher frequency, Q value, and insertion loss.

The bandwidth search is executed from the reference point. The active marker or the maximum trace value can be selected as the reference. The bandwidth search function detects lower and higher cutoff frequencies that differ from the reference point response by a user-specified bandwidth value (usually – 3 dB).



Bandwidth Search

Bandwidth parameters

Parameter Description	Symbol	Definition	Formula
Bandwidth	**BW**	The difference between the higher and lower cutoff frequencies.	F2 – F1
Center Frequency	**cent**	The midpoint between the higher and lower cutoff frequencies.	(F1+F2)/2
Lower Cutoff Frequency	**low**	The lower frequency point of the intersection of the bandwidth cutoff level and the trace.	F1
Higher Cutoff Frequency	**high**	The higher frequency point of the intersection of the bandwidth cutoff level and the trace.	F2
Quality Factor	**Q**	The ratio of the center frequency to the bandwidth.	cent/BW
Loss	**loss**	The trace measured value in the reference point of the bandwidth search.	—

To enable/disable bandwidth search function:

Search > Bandwidth/Notch Search

And use the toggle to turn it ON/OFF.

CALC:MARK:BWID	Turns the bandwidth search function ON/OFF.

Set the bandwidth search type by:

Search > Bandwidth/Notch Search > Search [Band | Notch]

CALC:MARK:BWID:TYPE	Sets the type of the bandwidth search function.

To set the bandwidth search level:

Search > Bandwidth/Notch Search> Level

Then either type in the desired level or use the arrows in the textbox.

To set the search reference:

Search > Bandwidth/Notch Search > Search Ref [Peak | Marker]

To select the active marker being searched:

Search > Bandwidth/Notch Search > Active Marker

Then use the dropdown menu to select the desired marker.

Flatness

The flatness search function allows to determine and view the following trace parameters: gain, slope, and flatness. Two markers to specify the flatness search range should be set.

Flatness Search

Flatness parameters

Parameter Description	Symbol	Definition
Gain	**gain**	Marker 1 value.
Slope	**slope**	Difference between marker 2 and marker 1 values.
Flatness	**flat**	Sum of “positive” and “negative” peaks of the trace, which are measured from the line connecting marker 1 and marker 2 (See above figure).

To enable/disable the flatness search function:

Search > Flatness Evaluation

Then use the toggle to turn it ON/OFF.

CALC:MARK:MATH:FLAT:STAT	Turns the marker flatness function ON/OFF.

To select the markers specifying the flatness search range:

Search > Flatness Evaluation > Start

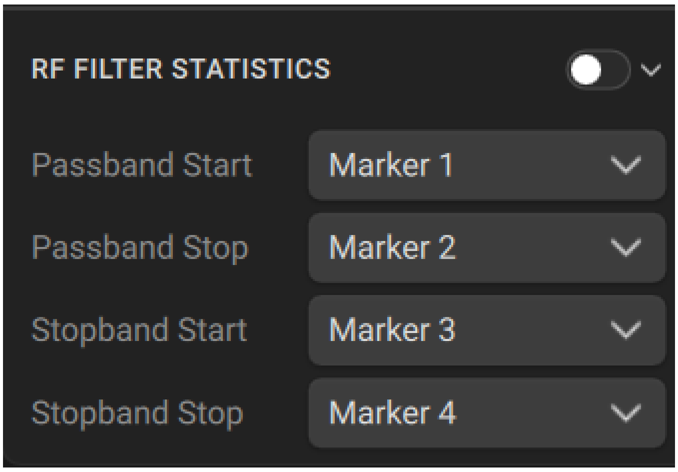
Search > Flatness Evaluation > Stop

Use the dropdown menus to select the desired start and stop markers.

CALC:MARK:MATH:FLAT:DOM:STAR	Sets or reads out the number of the marker, which specifies the start frequency of the flatness function domain.
CALC:MARK:MATH:FLAT:DOM:STOP	Sets or reads out the number of the marker, which specifies the stop frequency of the flatness function domain.

RF Filter Statistics

The RF filter statistics function allows to determine and view the following filter parameters: loss, peak-to-peak in a passband, and rejection in a stopband. The passband is specified by the first pair of markers, and the stopband is specified by the second pair of markers (See figure below).



RF filter statistics

RF filter statistics parameters

Parameter Description	Symbol	Definition
Loss in passband	**loss**	Minimum value in the passband.
Peak-to-peak in passband	**p-p**	Difference between maximum and minimum in the passband.
Reject	**rej**	Difference between maximum in stopband and minimum in passband.

To enable/disable the RF filter statistics function:

Search > RF Filter Statistics

Then use the toggle to turn it ON/OFF.

To select the markers specifying the passband:

Search > RF Filter Statistics > Passband Start

Search > RF Filter Statistics > Passband Stop

To select the markers specifying the stopband:

Search > RF Filter Statistics > Stopband Start

Search > RF Filter Statistics > Stopband Start

Memory Trace Function

An associated memory trace can be created for each data trace. The memory trace is saved at the moment when the corresponding softkey is pressed or a program command is received. After saving the memory trace, the screen displays two traces — data and memory. The following settings of the memory and traces display can be performed:

Trace Display	Trace status field
Data and memory	**D&M**
Memory only	**M**
Data only	**Dat**
Data and memory OFF	**Off**

NOTE Up to 8 memory traces can be created for each data trace. For a detail description, see Memory FIFO.

The memory trace is displayed in the same color as the main data trace, but it is half as bright (color and brightness of data and memory traces can be customized, see User Interface Setting).

The memory trace is used for displaying and mathematical operations with data trace. For a detail description, see Mathematical Operations.

In fact, complex measurement data is saved in memory, not their graphical representation. Consequently:

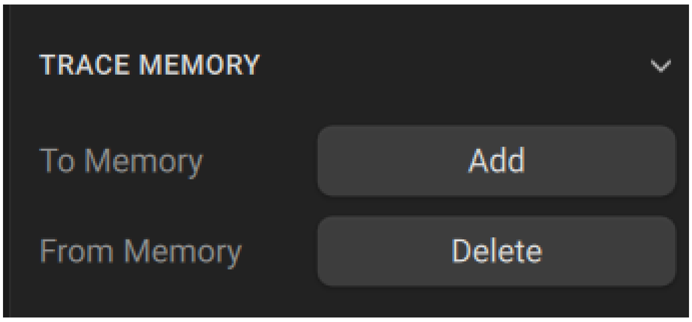
- Mathematical operations are carried out between the current and stored S-parameters.
- The memory trace changes similar to an associated data trace when the settings are changed, such as Format, Electrical delay, Time domain, etc.

NOTE The memory trace cannot be extrapolated or interpolated, so when the frequency range or sweep type are changed, the memory contents become incorrect. When the number of points is changed, the memory is automatically cleared.

Saving Data Trace into Memory

The function of saving data traces into memory is applied to an individual trace or to all traces of the channel at once.

The trace to which the function is applied must be preselected as active (See Selection of Active Trace/Channel).



Trace Memory

To save an active data trace into the memory, use the following softkeys:

Memory > To Memory > Add

CALC:MATH:MEM	Copies the measurement data to the memory trace.

Erasing Memory

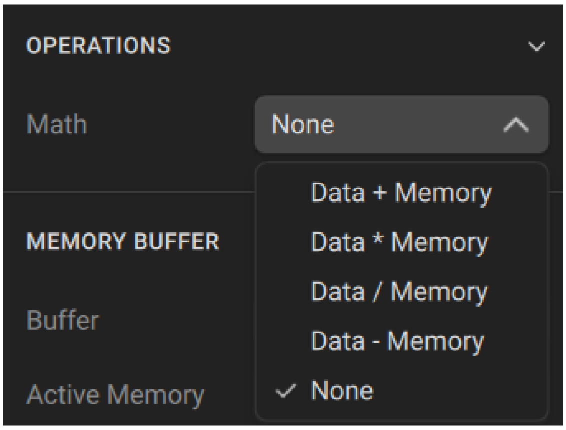
The trace to which the function is applied must be preselected as active (See Selection of Active Trace/Channel).

To erase the memory of the active trace:

Memory > From Memory > Delete

Trace Display Setting

The trace to which the function is applied must be preselected as active (See Selection of Active Trace/Channel).



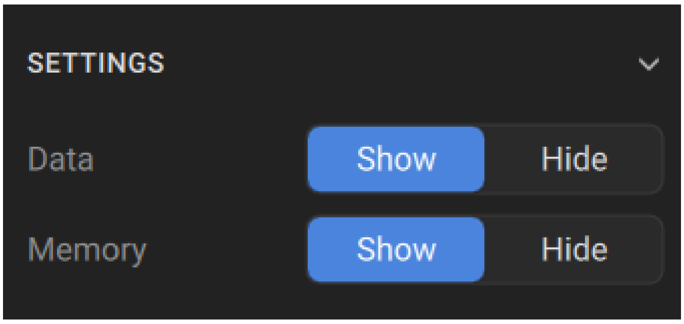
Trace Data Display via Memory Operations Menu

To set the type of traces to be displayed on the screen:

Memory > Operations > Math [Data + Memory | Data * Memory | Data / Memory | Data - Memory | None]

DISP:WIND:TRAC:MEM	Turns the memory trace display ON/OFF.
DISP:WIND:TRAC:STAT	Turns the data trace display ON/OFF.

Memory Settings



To show or hide trace data or trace memory:

Memory > Settings > Data [Show | Hide]

Memory > Settings > Memory [Show | Hide]

Memory Buffer

The memory buffer function increases the number of memory traces up to 8 for each data trace. Memory traces are saved in a buffer queue.

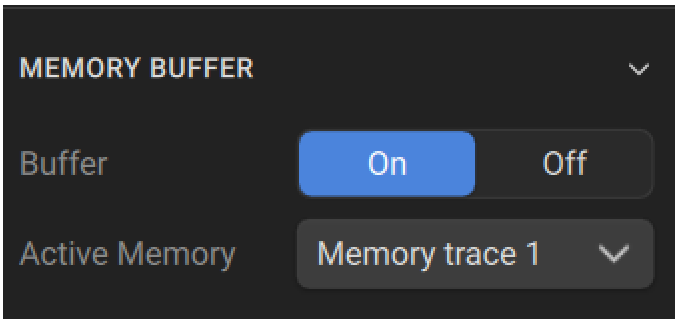
By default, the memory buffer function is disabled, the queue depth is 1, so there is only 1 memory trace associated with each data trace. It is Memory Trace Function.

When the memory buffer function is enabled, the queue depth increases to 8, so it is possible to record up to 8 memory traces for each data trace.

Memory traces are saved in chronological order. The new memory trace is numbered 1, and the numbers of the previous memory traces are increased by one. If the number of memory traces in the memory buffer exceeds 8, the oldest trace is discarded.

All memory traces contained in the memory buffer are displayed simultaneously.

For math operations, only one of memory buffer trace is used (such a trace is called active). By default, the newest memory trace is active. If necessary, any trace in the memory buffer can be activated.



Memory Buffer Menu

To enable / disable the function of saving to memory buffer:

Memory > Memory Buffer > Buffer [ON/OFF]

To assign a memory trace as active for math operations:

Memory > Memory Buffer > Active Memory> Memory trace [1 | 2 | 3 ... 8]

The memory traces in the buffer are arranged in chronological order, where 1 is the newest save, 8 is the oldest.

Mathematical Operations

The memory trace can be used for mathematical operations with the data trace. The mathematical operations are performed on complex values before they are formatted for display. The result of math operation replaces the data trace. The following mathematical operations can be performed:

Data/ Memory	Divides the measured data by the memory data. The trace status field indicates: **D/M**.
Data* Memory	Multiplies the measured data by the memory data. The trace status field indicates: **D*M**.
Data– Memory	Subtracts a memory data from the measured data. The trace status field indicates: **D–M**.
Data+ Memory	Adds the measured data to the memory data. The trace status field indicates: **D+M**.
Normalization	Pressing the **Normalization** softkey performs 3 steps in sequence: 1. Saves the current data into memory. 2. Turns on the math operation **Data/ Memory** (normalizes the measured data). 3. Turns on "data only" display type. The trace status field indicates: **D/M Dat.**

To access math operations:

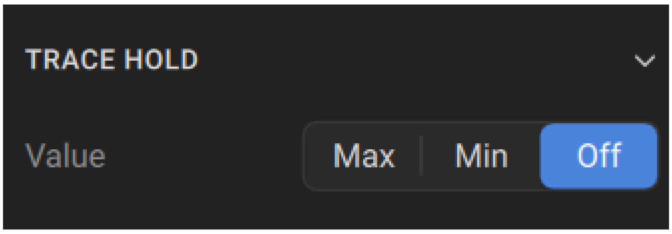
Memory > Operations > Math [Data + Memory | Data * Memory | Data / Memory | Data - Memory | None]

CALC:MATH:FUNC	Selects the math operation between the measured data and the memory data.

NOTE	If the memory buffer function is turned on, check if the active memory trace is the desired trace for math operation (See Memory Buffer).

Trace Hold

The trace hold function is used to hold the maximum or minimum values of the trace.



To turn ON/OFF trace hold function:

Trace > Trace Hold > Value [Max | Min | Off]

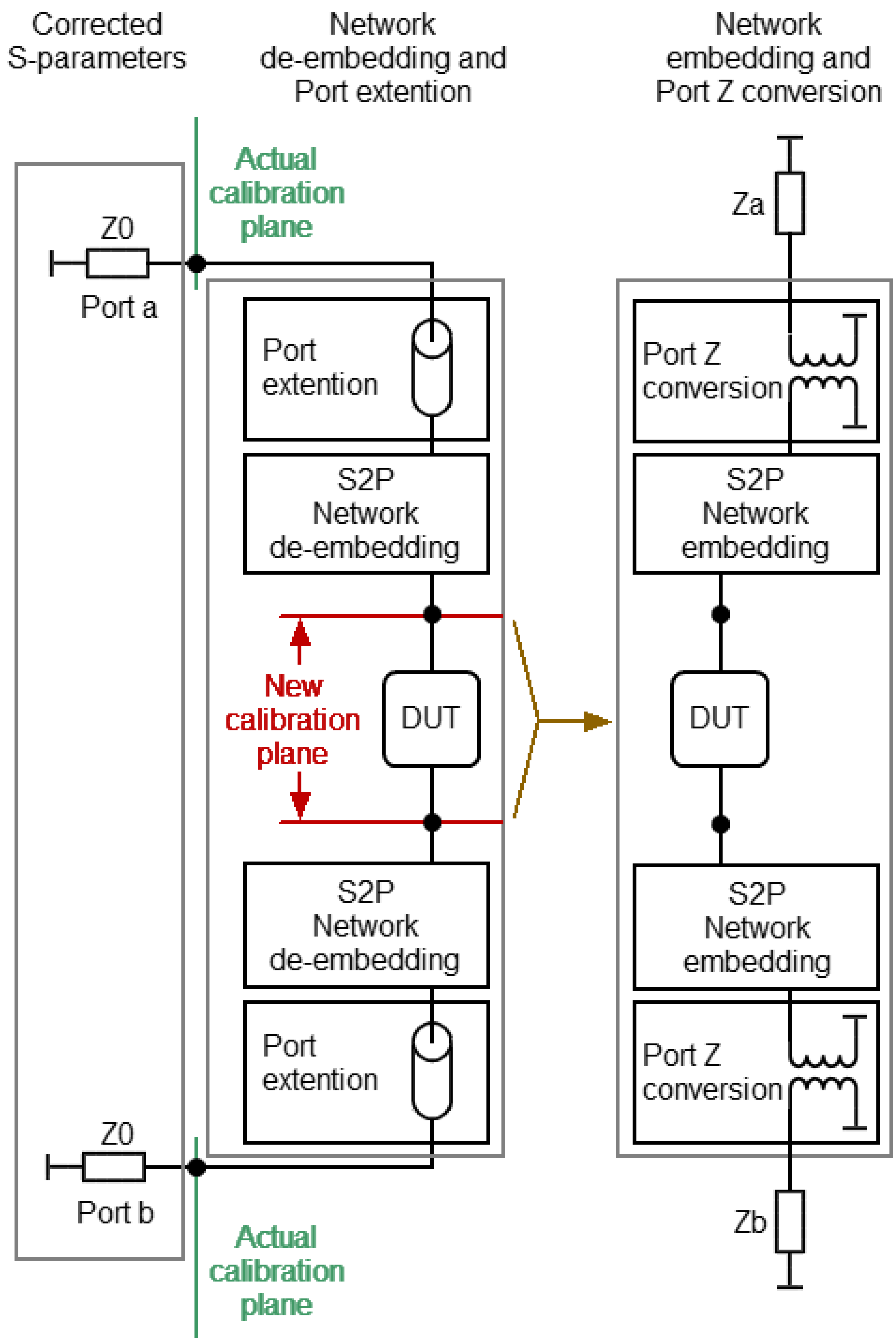
CALC:HOLD:TYPE	Sets the type of the trace hold function.
CALC:HOLD:CLEar	This command resets the trace hold function.

Fixture Simulation

The fixture simulation functions are a set of software functions for mathematically simulating measurement conditions that are different from the actual measurement conditions. The following conditions can be simulated:

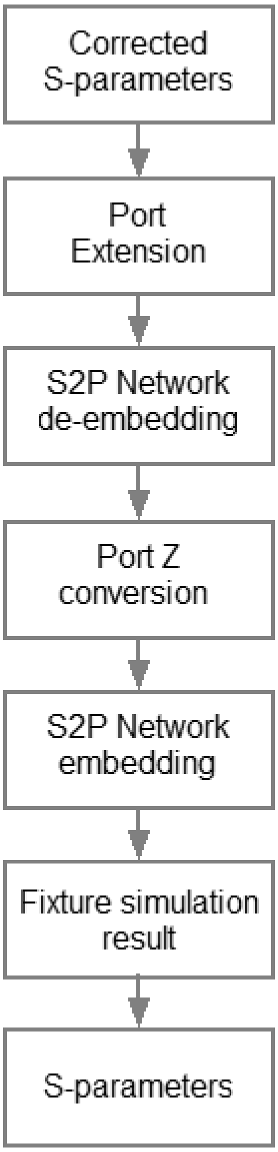
- Port Extension
- Port reference impedance conversion
- Circuit de-embedding
- Circuit embedding

The logic diagram of the fixture simulation function is shown in the figure below.



Logic diagram of fixture simulation function

The data processing flow diagram of the fixture simulation feature is shown in the the figure below.

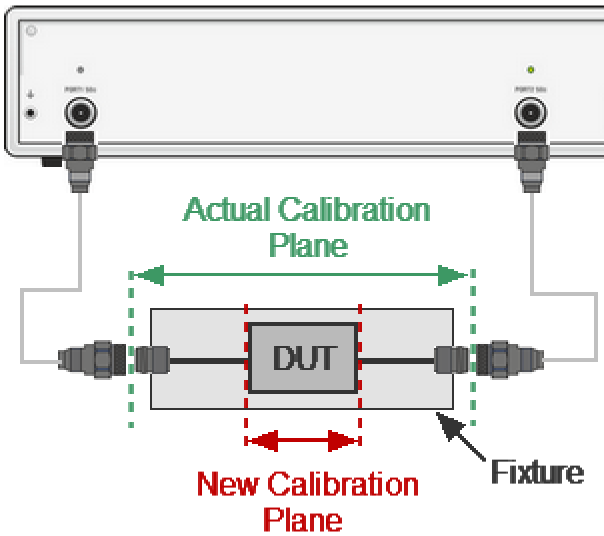


Data processing flow diagram of fixture simulation function

The channel to which the function is applied must be preselected as active (See Selection of Active Trace/Channel). Fixture simulation functions affect all the traces of the channel.

Port Extension

The port extension function moves the calibration plane toward the DUT terminals by the specified electrical delay value. The function is useful when a fixture is used for the DUT connecting and the calibration cannot be performed at the DUT terminals. The calibration plane can be established at coaxial connectors of the fixture and then moved to the DUT terminals using the port extension function (See figure below).



Port extension

The function uses the model of the perfectly matched transmission line with loss with parameters:

- The phase incursion in the line

$$\Delta\varphi = e^{-j \cdot 2\pi \cdot f \cdot \tau},$$

  where f — frequency, Hz,

  τ — electrical delay, sec.

- The loss of the line $L(f)$ can be specified by one of the following methods:

  1. Frequency-independent loss at DC (L_0)

  $$L(f) = L_0.$$

  2. Loss determined by the losses in two frequency points (L_0 at DC, and L_1 at frequency F_1)

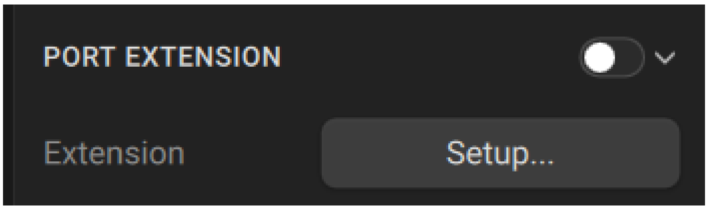
  $$L(f) = L_0 + (L_1 - L_0)\sqrt{\frac{f}{F_1}}$$

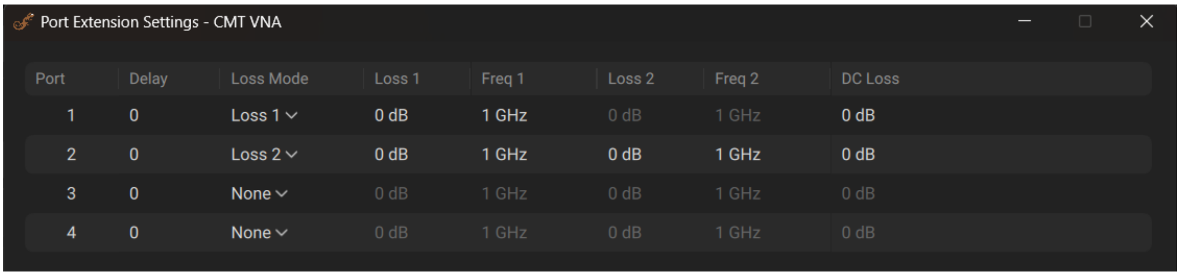
3. Loss determined by the losses in three frequency points (L_0 at DC, L_1 at frequency F_1 and L_2 at frequency F_2)

$$L(f) = L_0 + (L_1 - L_0)(\frac{f}{F_1})^n ,$$

$$n = \frac{log|\frac{L_1}{L_2}|}{log\frac{F_1}{F_2}} .$$

NOTE The accuracy of the port extension method depends on the fixture used. The closer the fixture parameters are to the model of a perfectly matched transmission line, the higher the accuracy is.





Port	Delay	Loss Mode	Loss 1	Freq 1	Loss 2	Freq 2	DC Loss
1	0	Loss 1	0 dB	1 GHz	0 dB	1 GHz	0 dB
2	0	Loss 2	0 dB	1 GHz	0 dB	1 GHz	0 dB
3	0	None	0 dB	1 GHz	0 dB	1 GHz	0 dB
4	0	None	0 dB	1 GHz	0 dB	1 GHz	0 dB

Port Extension Menu and Popup

To enable the port extension function:

Fixture > Port Extension

Then use the toggle to turn it ON/OFF.

SCPI SENSe:CORRection:EXTension

To set the electrical delay for each port:

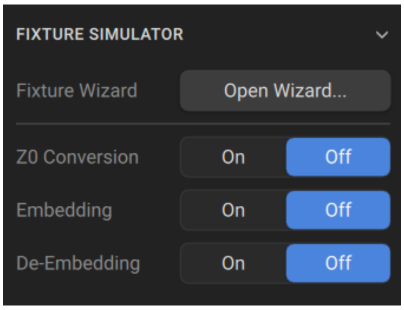
Fixture > Port Extension > Setup... > Delay

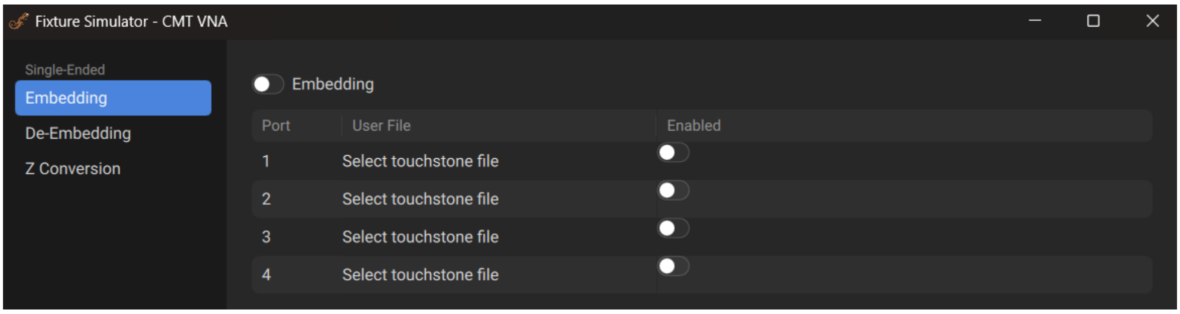
SCPI SENSe:CORRection:EXTension:PORT:TIME

	To select the loss: **Fixture > Port Extension > Setup... > Loss Mode [None |Loss 1| Loss 2]**
	Enter the L_1, F_1 values and enable the use of these values in further calculations: **Loss Mode** **Loss1** **Frequency 1** Perform the same steps for L_2, F_2.
	Enter the L_0 value: **DC Loss**
SCPI	SENSe:CORRection:EXTension:PORT:INCLude SENSe:CORRection:EXTension:PORT:FREQuency SENSe:CORRection:EXTension:PORT:LOSS SENSe:CORRection:EXTension:PORT:LDC

Fixture Simulation

CMT VNA offers a Fixture Simulation Wizard to allow the user to use embedding, de-embedding, and Z0 conversion.





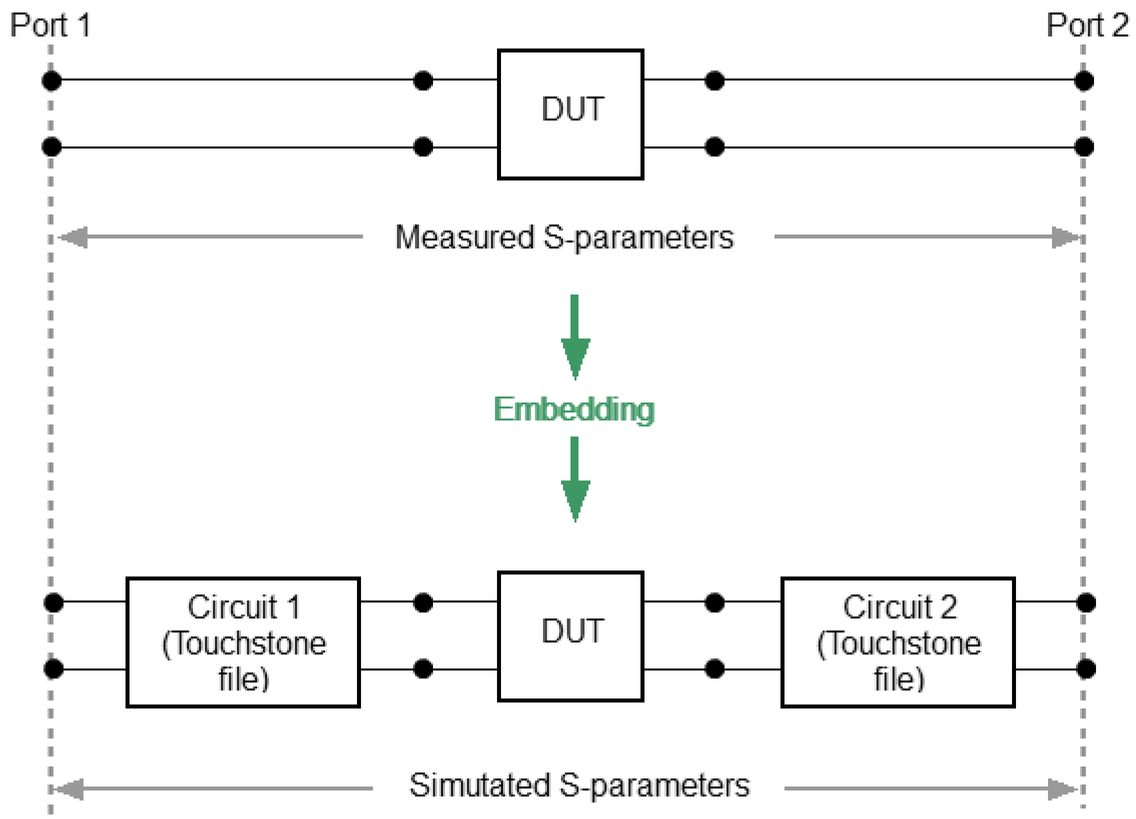
Fixture Simulation Menu and Wizard

Embedding

Embedding is a function of the S-parameter transformation via integration of some virtual circuit into the real network (See figure below).

The embedding function allows to mathematically simulate the DUT parameters after adding the fixture circuits.

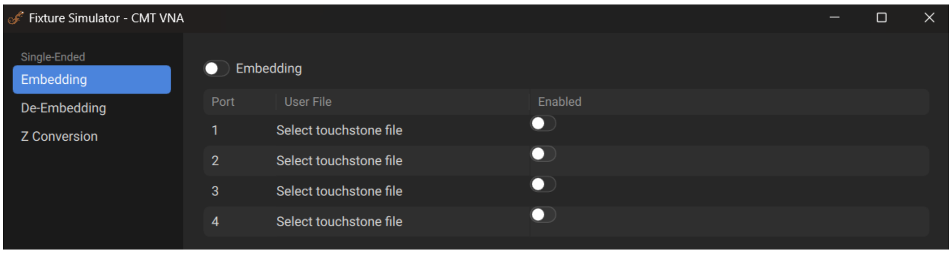
The embedding function is an inverted de-embedding function.



Embedding

The circuit being integrated should be defined in the data file containing S-parameters of that circuit. The circuit should be described as a two-port in Touchstone file (extension *.S2P), which contains the S-parameter table: S11, S21, S12, S22 for a number of frequencies.

NOTE	The S-matrices of all embedding circuits are oriented so that the S11 is directed to the Analyzer port and S22 directed to the DUT.



Fixture Simulation Wizard Embedding Menu

To enable/disable the embedding function, use the following softkeys:

Fixture > Embedding [ON|OFF]

Toggle on the embedding toggle at the top, then enable and select the touchstone file for each port.

Select the file by clicking on Select touchstone file.

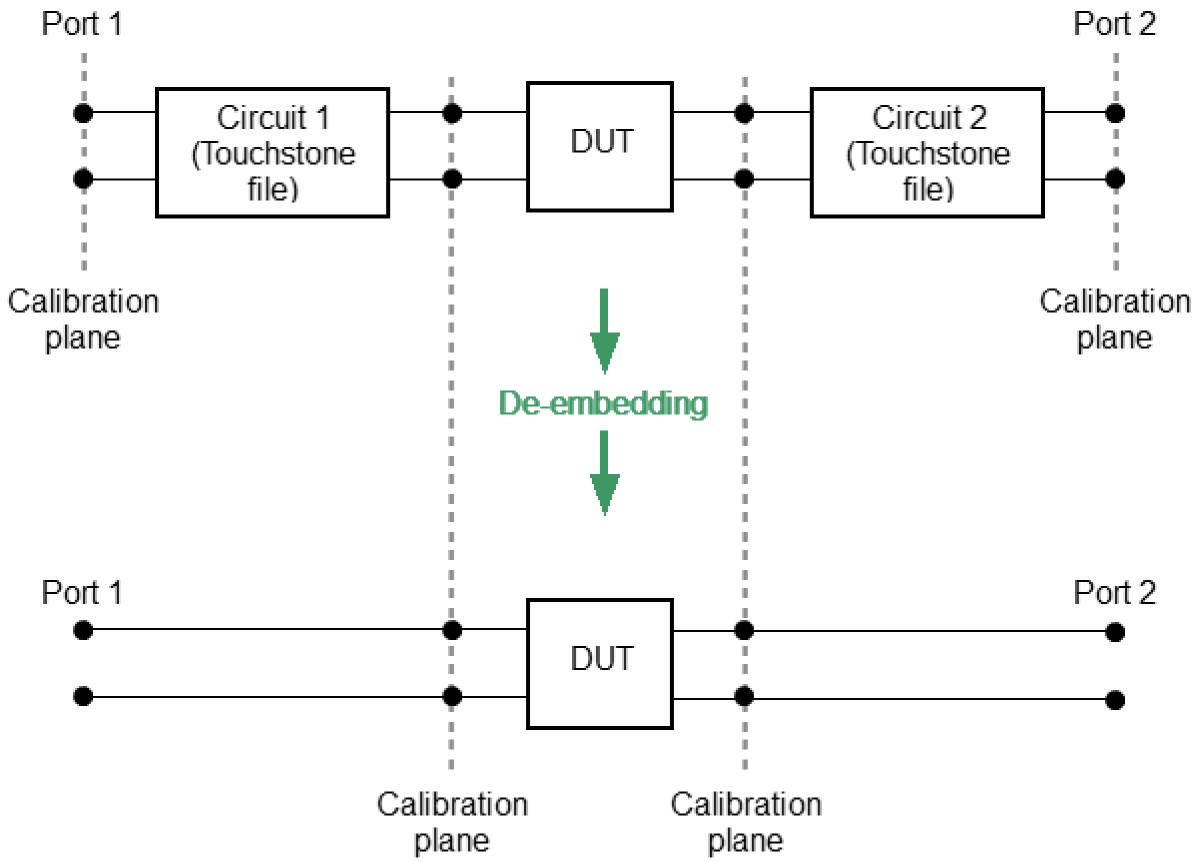
Enable the file by using the toggle under Enabled.

De-embedding

De-embedding is a function of transforming the S-parameter by eliminating some circuit effect from the measurement results.

The de-embedding function allows to mathematically exclude the effect of the fixture circuit existing between the calibration plane and the DUT in the real network from the measurement results. The fixture is used for the DUTs, which cannot be directly connected to the test ports.

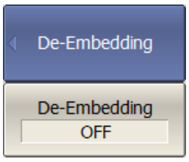
The de-embedding function shifts the calibration plane closer to the DUT, so as if the calibration has been executed on the network with this circuit removed (See figure below).



De-embedding

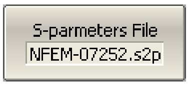
The circuit being removed should be defined in the data file containing S-parameters of that circuit. The circuit should be described as two-port in Touchstone file (extension *.S2P), which contains the S-parameter table: S11, S21, S12, S22 for a number of frequencies.

NOTE	The S-matrices of all de-embedding circuits are oriented so that the S11 is directed to the Analyzer port and S22 directed to the DUT.



To enable/disable the de-embedding function, use the following softkeys:

Analysis > Fixture Simulator > De-Embedding > De-Embedding [ON | OFF]



If the S-parameters file is not specified, the softkey for Port n activation will be grayed out.

To enter the file name of the de-embedded circuit S-parameters of Port n, use the following softkeys:

Analysis > Fixture Simulator > De-Embedding > S-parameters File

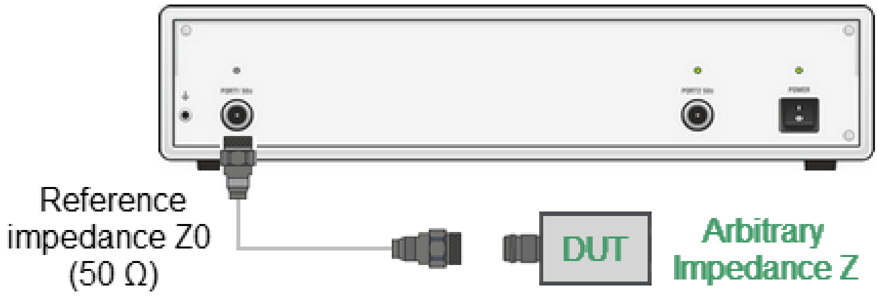
To enable/disable the de-embedding function for Port n, use the following softkeys:

Analysis > Fixture Simulator > De-Embedding > Port n [ON | OFF]

CALC:FSIM:SEND:DEEM:STAT	Turns the two-port network de-embedding function ON/OFF.
CALC:FSIM:SEND:DEEM:PORT:USER:FIL	Sets or reads out the name of the *.S2P file of the de-embedded circuit of the two-port network de-embedding function.
CALC:FSIM:SEND:DEEM:PORT:STAT	Turns the two-port network de-embedding function for specified port ON/OFF.

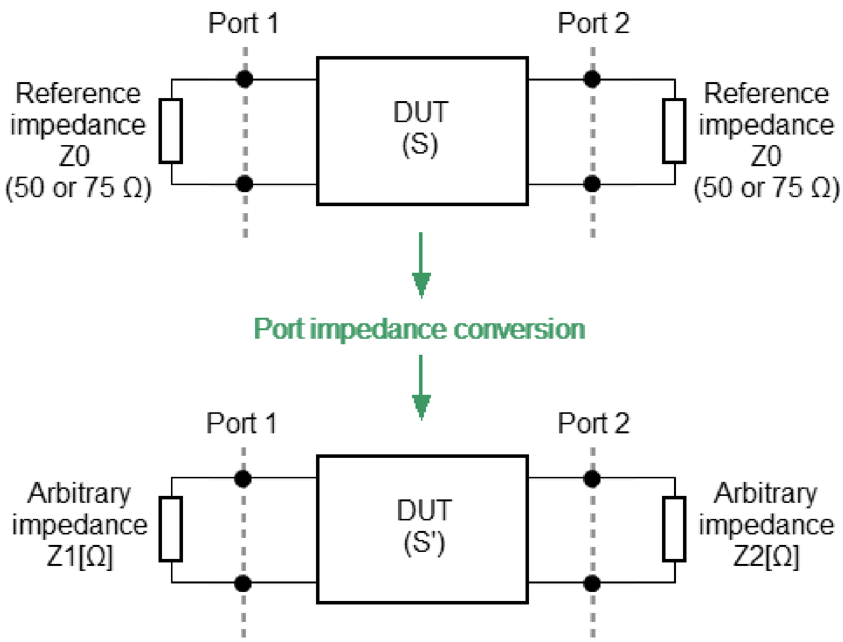
Port Reference Impedance (Z) Conversion

The default reference impedance of a port is equal to the impedance of the connectors (50 or 75 Ω). But in the process, it is often required to measure DUT with arbitrary resistance (See example in the figure below), not equal to the impedance of a port. In this case, it is possible to convert the reference impedance to an arbitrary impedance value using the software.



Example of measuring a DUT with an arbitrary impedance by the Analyzer with reference impedance 50 Ω

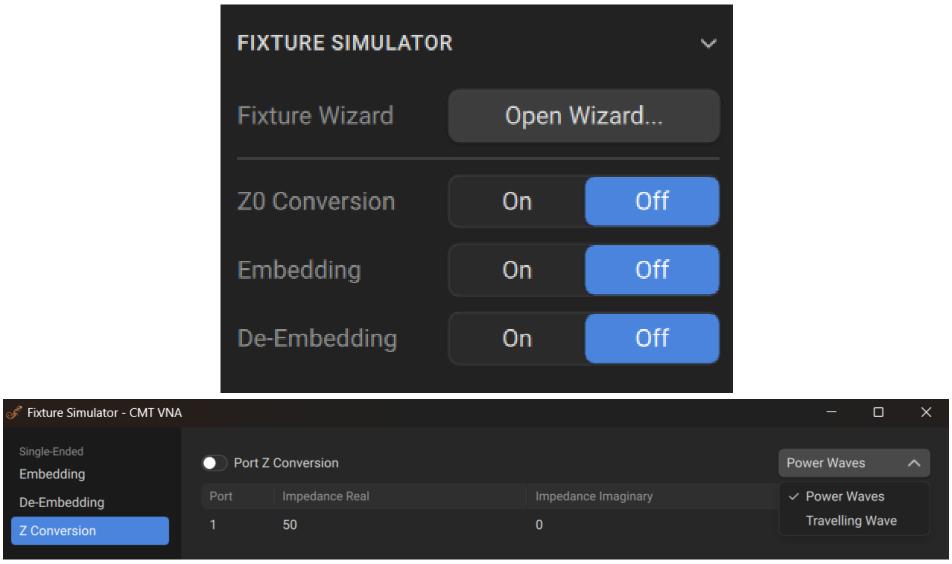
Port reference impedance conversion is a function that mathematically converts the matrix of S-parameters measured at the reference impedance of port Z0 to the matrix of S-parameters measured at an arbitrary impedance of port Z1 (See figure below). The function is also referred to as the renormalization transformation of S-parameters.



Port reference impedance conversion

Renormalization can be based on two alternative microwave circuit theories, whose conversion formulas may yield different results if the reference impedance of at least one test port has a non-zero imaginary part. The first theory is "A General Waveguide Circuit Theory" (R.B.Marks and D.F.Williams). The second theory is the "Power waves and the Power Scattering Matrix" (K.Kurokawa).

NOTE	The source value of the Z0 port reference impedance (commonly 50 Ω) is defined in the process of the calibration. It is determined by the characteristic impedance of the calibration kit and its value.



Fixture Simulation Menus

To enable/disable the port reference impedance conversion function:

Fixture > Z0 Conversion [ON|OFF]

Or go into the Fixture Wizard and go to the Z Conversion tab.

CALC:FSIM:SEND:ZCON:STAT	Turns the port impedance conversion function ON/OFF.

To enter the value of the simulated impedance of Port n:

Fixture > Fixture Wizard > Z Conversion > Impedance Real

Fixture > Fixture Wizard > Z Conversion > Impedance Imaginary

CALC:FSIM:SEND:ZCON:PORT:Z0	Sets or reads out the value of the impedance of the port impedance conversion function.
CALC:FSIM:SEND:ZCON:PORT:Z0:REAL	Sets or reads out the real part of the impedance of the port impedance conversion function.
CALC:FSIM:SEND:ZCON:PORT:Z0:IMAG	Sets or reads out the imaginary part of the impedance of the port impedance conversion function.

To choose the theory according to which the renormalization of S-parameters is performed:

Fixture > Fixture Wizard > Z Conversion

Use the dropdown menu in the top right to select either Power Waves or Travelling Waves.

CALC:FSIM:SEND:ZCON:THE	Sets or reads out the waveguide circuit theory used to renormalize the S-parameters.

Time Domain Transformation

NOTE	The availability of this feature depends on the Analyzer model (See corresponding datasheet).

The Analyzer measures parameters of the DUT in the frequency domain. Time domain transformation is a function of mathematical transformation of the measured parameters in order to obtain the time domain representation.

Time domain function simulates time-domain reflectometry. The meaning of which is to influence the DUT with a pulsed or step signal, followed by the analysis of the reflected signal. The magnitude, duration, and shape of the reflected signal determine the nature of the impedance variation in the DUT. The Analyzer does not affect the DUT either in pulses or steps. Instead, a Chirp-Z transform algorithm is used to calculate time information from the frequency measurements. The Chirp-Z transform is a generalization of the Fourier transform that allows to set arbitrary transform start and stop values.

Transformation Types

The time domain function supports the following transformation types:

- **Bandpass** mode simulates the response of the bandpass network to the impulse.
- **Lowpass impulse** mode simulates the response of the lowpass network to the impulse.
- **Lowpass step** mode simulates the response of the lowpass network to the unit step function.

The time domain resolution in the lowpass mode is twice as high as in the bandpass mode. The bandpass mode determines the distance to the discontinuity but does not provide information about the nature of the discontinuity. The lowpass mode determines the distance to the discontinuity and provides information about the nature of the discontinuity (open or short circuit, for example). The lowpass step mode is useful for the impedance measurement along the distance.

Bandpass mode is applied to the DUTs that do not operate with DC current such as band pass filters. The frequency settings in the bandpass mode can be arbitrary.

Lowpass mode is applied to the DUTs that operate with DC current such as cables.

The frequency settings in the lowpass mode is required to be a harmonic frequency grid, where the frequency value at each frequency point is an integer multiple of the start frequency. The Analyzer can set the harmonic frequency grid from the current frequency settings with one click.

The value of the DUT response at DC is required to be known in the lowpass mode. The DC value cannot be measured directly by the Analyzer. The Analyzer offers two options: the DC value is automatically extrapolated or manually set. The last option is used when the DUT response at DC is well known, for example, for a low loss cable the DC value is:

- "1" for open-ended cable.
- "-1" for a short-circuited cable.
- "0" for a cable terminated with a matched load.

Transformation Unambiguity Range

The time domain response is a periodic function due to the discrete nature of the frequency response. The time domain unambiguity range is determined by the step in the frequency domain:

$$\Delta T = \frac{1}{\Delta F}; \quad \Delta F = \frac{F_{max} - F_{min}}{N-1}$$

Windowing

The time domain response has a ringing due to the finite nature of the frequency response. To reduce the ringing the windowing is applied to the frequency response. The time domain transformation function applies the Kaiser window function. The window function selection is a tradeoff between the ringing reducing and the time domain resolution.

The Kaiser window is defined by the β parameter, which smoothly fine-tunes the window shape from minimum (rectangular) to maximum. The user can fine-tune the window shape, or select one of the three pre-programmed windows:

- **Minimum** (rectangular)
- **Normal**
- **Maximum**

Pre-programmed window types

Window	Lowpass Impulse		Lowpass Step	
	Side Lobes Level	Pulse Width	Side Lobes Level	Edge Width
Minimum	– 13 dB	$\frac{0.6}{F_{max}-F_{min}}$	– 21 dB	$\frac{0.45}{F_{max}-F_{min}}$
Normal	– 44 dB	$\frac{0.98}{F_{max}-F_{min}}$	– 60 dB	$\frac{0.99}{F_{max}-F_{min}}$
Maximum	– 75 dB	$\frac{1.39}{F_{max}-F_{min}}$	– 70 dB	$\frac{1.48}{F_{max}-F_{min}}$

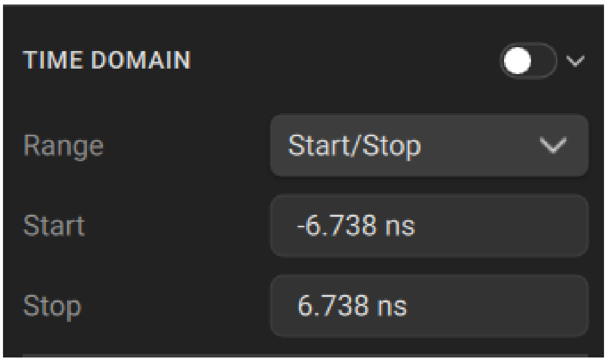
X-axis Representation

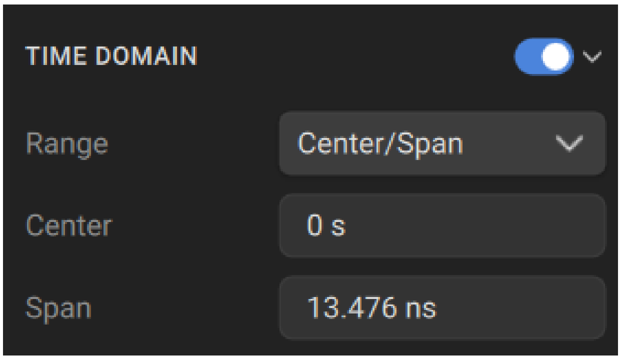
The X-axis units can be set in seconds or distance units (meters or feet). When the distance units are selected, the velocity factor is used to compute the distance from time. The velocity factor setting is located in the cable correction function (See Cable Correction Function).

The two types of reflection can be selected: round trip or one way. The round trip setting shows the total time or distance that the signal travels in both directions along the DUT. The one-way setting shows the time or distance the signal travels in one direction along the DUT.

NOTE	As the time domain transformation can be applied for separate traces of a channel, the x-axis units and round trip/one-way type depends on the active trace selected.

The time domain transformation is applied for separate traces of a channel. The trace to which the function is applied must be preselected as active (See Selection of Active Trace/Channel).





Time Domain Transformation Activation

To enable/disable time domain transformation function:

Time Domain > Time Domain

And then use the toggle to turn it ON/OFF.

CALC:TRAN:TIME:STAT	Turns the time domain transformation function ON/OFF.

NOTE	Time domain transformation function is accessible only in linear frequency sweep mode.

Time Domain Transformation Span

To define the span of time domain representation, its start and stop, or center and span values can be set.

To set the start and stop limits of the time domain range:

Time Domain > Time Domain > Start

Time Domain > Time Domain > Stop

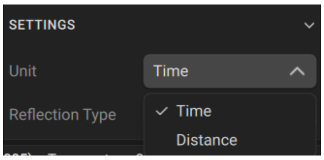
CALC:TRAN:TIME:STAR	Sets or reads out the time domain start value when the time domain transformation function is turned ON.
CALC:TRAN:TIME:STOP	Sets or reads out the time domain stop value when the time domain transformation function is turned ON.

To set the center and span of the time domain:

Time Domain > Time Domain > Center

Time Domain > Time Domain > Span

CALC:TRAN:TIME:CENT	Sets or reads out the time domain center value when the time domain transformation function is turned ON.
CALC:TRAN:TIME:SPAN	Sets or reads out the time domain span value when the time domain transformation function is turned ON.

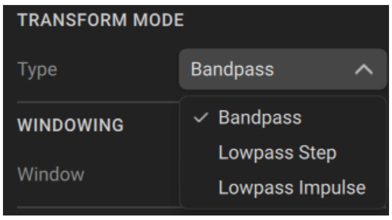


To set the unit of the time domain:

Time Domain > Settings > Unit [Time | Distance]

CALC:TRAN:TIME:UNIT	Selects the transformation unit for the time domain transformation function: seconds, meters, feet.

Time Domain Transformation Type



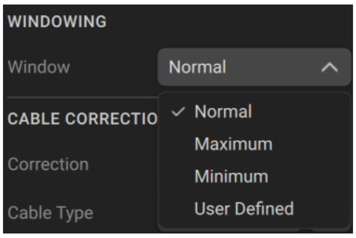
To set the time domain transformation type:

Time Domain > Transform Mode > Type [Bandpass | Lowpass Step | Lowpass Impulse]

CALC:TRAN:TIME	Selects the transformation type for the time domain transformation function: band-pass or low-

	pass.
CALC:TRAN:TIME:STIM	Selects the stimulus type for the time domain transformation function: impulse or step.

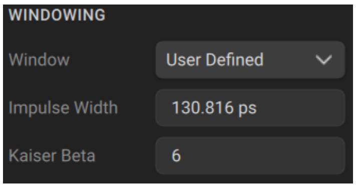
Time Domain Transformation Window Shape Setting



To set the window shape, use the following softkeys:

Time Domain > Windowing

Then use the dropdown to select the type: Normal, Maximum, Minimum, or User Defined.



To set the window shape for the specific impulse width or front edge width:

Time Domain > Windowing > Window: User Defined > Impulse Width

Type in the desired width or use the arrows in the textbox.

CALC:TRAN:TIME:IMP:WIDT	Sets or reads out the impulse width (time domain transformation resolution), coupled with the Kaiser-Bessel window shape β parameter.
CALC:TRAN:TIME:STEP:RTIM	Sets or reads out the rise time of the step signal (time domain transformation resolution), coupled with the Kaiser-Bessel window shape β parameter.

To set the window shape for the specific β-parameter of the Kaiser-Bessel filter:

Time Domain > Windowing > Window: User Defined > Kaiser Beta

The available β values are from 0 to 13:

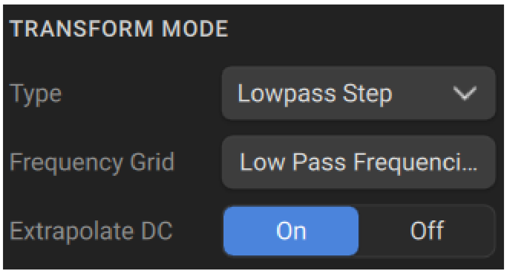
- "0" corresponds to minimum window.
- "6" corresponds to normal window.
- "13" corresponds to maximum widow.

CALC:TRAN:TIME:KBES	Sets or reads out the β parameter, which controls the Kaiser-Bessel window shape when performing the time domain transformation.

NOTE	The impulse width and β of the Kaiser-Bessel filter are the dependent parameters. When setting one of the parameters the other one will be adjusted automatically.

Cable Correction Settings

When the length units are selected the velocity factor setting of the Cable correction function affects the X-axis scale. See Cable Correction Function.



Lowpass Mode Settings

If lowpass mode is used, the frequency range must be set to a harmonic grid. The frequency values in measurement points are integer multiples of the start frequency.

In lowpass mode, the value of the DUT response at DC is extrapolated from the first few frequency points, or manually set. Set the DC value manually if the response of DUT is well known. For example, if the DUT is a cable then DC value is:

- "1" for open-ended cable.
- "-1" for a short-circuited cable.
- "0" for a cable terminated with a matched load.

To create a harmonic grid for the current frequency range:

Time Domain > Transform Mode > Frequency Grid

CALC:TRAN:TIME:LPFR	Changes the frequency range to match with the lowpass type of the time domain transformation

function.

To turn on/off the automatic extrapolation of DC value:

Time Domain > Transform Mode > Extrapolate DC [ON | OFF]

CALC:TRAN:TIME:EXTR:DC	Turns ON/OFF the DC extrapolation, when the time domain transformation function is turned ON.

To set the DC value manually (when Extrapolate DC is off):

Time Domain > Transform Mode > DC Value

CALC:TRAN:TIME:DC:VAL	Sets or reads out the DC value used in the lowpass type of the time domain transformation, when the DC extrapolation is OFF.

NOTE

The **Set Frequency Low Pass**, **Extrapolate DC**, **DC Value** softkeys are duplicated in the Gating menu. The settings they make have the same effect on Time Domain and Gating.

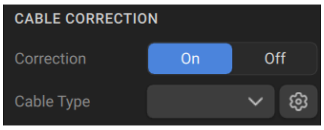
The **Type** (of time domain transformation) softkey is related to the **DUT Low Pass** softkey in the Gating menu as follows:

- If **Type** set to **Lowpass [Impulse or Step]**, **DUT Low Pass** turn ON.
- If **Type** set to **Bandpass**, **DUT Low Pass** turn OFF.

Cable Correction Function

Cable correction function allows to consider the influence of cable characteristics during transform in the time domain. The function contains the cable velocity factor and the cable loss in dB/m. The cable loss value is indicated at the specified frequency. All values can be set manually or selected from the table of predefined cables. The velocity factor is used to convert the time units to the distance units. The cable loss value, together with the frequency, are used to compensate for the attenuation in the cable, so that, for example, the response to an open circuit is unity. The cable correction function is disabled by default.

Cable Correction Activation



To enable/disable cable correction function of the time domain transformation function, use the following softkeys:

Time Domain > Cable Correction > Correction [ON | OFF]

SENS:CORR:TRAN:TIME:STAT	Turns the cable correction ON/OFF when the time domain transformation function is turned ON.

Cable Table

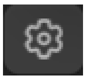
The software contains the predefined table of cables (See figure below). Each row of the table contains the cable name and the following parameters: velocity factor, cable loss and frequency. The cable type can be selected either via the dropdown menu where you can scroll through the list of available cables, or click on the gear button in the Cable Type section to access the full cable table.

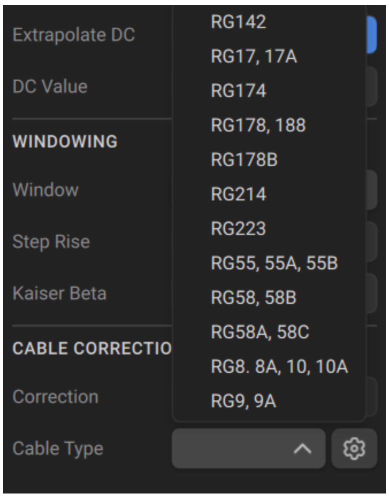
All table fields can be edited. Changes are saved automatically.

If there is no cable description in the table, it is possible to add it. To do this, create a new row in the table using the **Add New Cable** button and enter its name and parameters.

Cable Table

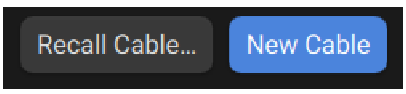
To open the cable table:

Time Domain > Cable Correction > Cable Type 

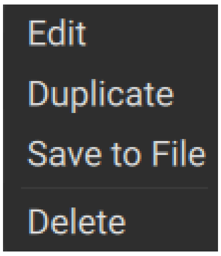


To select the cable:

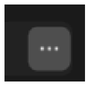
Time Domain > Cable Correction > Cable Type [select from dropdown menu]



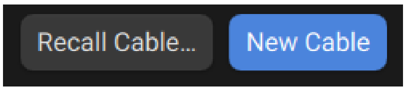
To add the new cable in the table, use the **Add New Cable** softkey:

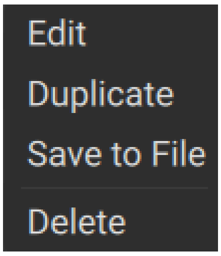


NOTE: A new cable can be added in the table by specifying its name and parameters in the empty field at the end of the table.

To delete cable from the table, use the 

that appears when you hover over the right-hand side of the cable in the cable table, then select Delete.





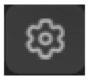
To save cable table in file, use the **Save to File...** softkey that appears when you hover over the right-hand side of the cable in the cable table.

To load cable table in file, use the **Recall Cable...** softkey.

Velocity Factor

Velocity factor is used to calculate distance along a cable from the cable delay value. If the cable correction function is disabled, the software assumes it to be equal to 1. To obtain the accurate mismatch location in a cable, it is important to set the right velocity factor of the cable.

To set the velocity cable:

Time Domain > Cable Correction > Cable Type  **> Velocity Factor**

SENS:CORR:TRAN:TIME:RVEL	Sets or reads out the cable relative wave speed velocity for the cable correction function, when the time domain transformation function is turned ON.

NOTE	The velocity factor value can also be set by the selecting the cable in the cable table.

Cable Loss

The cable loss value is used to compensate the signal attenuation in a cable. The cable loss value is set in dB per meter.

To set the cable loss:

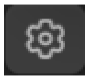
Analysis > Time Domain > Cable Correction > Cable Loss

SENS:CORR:TRAN:TIME:LOSS	Sets or reads out the cable loss value for the cable correction function when the time domain transformation function is turned ON.

NOTE	The cable loss value can also be set by the selecting the cable in the cable table.

Frequency

To set the frequency, at which the cable loss is specified, use the following softkeys:

Time Domain > Cable Correction > Cable Type  **> Loss**

SENS:CORR:TRAN:TIME:FREQ	Sets or reads out the frequency value at which the cable loss is specified for the cable correction function when the time domain transformation function is turned ON.

Time Domain Gating

NOTE	The availability of this feature depends on the Analyzer model (See corresponding datasheet).

Time domain gating is a function that mathematically removes unwanted responses in the time domain. The function performs a time domain transformation, selects the region in the time domain, deletes the response inside (or outside) the selected region and transforms back to the frequency domain. The function allows the user to remove spurious effects of the fixture in the frequency domain, if the useful signal and spurious signal are separable in the time domain.

The recommended procedure is as follows:

- Use the time domain function for viewing the layout of useful and spurious responses.
- Enable the time domain gating and set the gate position to remove as much of spurious response as possible.
- Disable the time domain function and view the response without spurious effects in frequency domain.

The function involves two types of time gate:

- **Bandpass** — removes the response outside the time gate span.
- **Notch** — removes the response inside the time gate span.

The sharp gate shape leads to ringing effect in the frequency domain. To reduce the ringing the gate shape can be smoothed. The following gate shapes are offered:

- **Maximum**
- **Wide**
- **Normal**
- **Minimum**

The minimum window has a sharp shape. The maximum window has a more smoothed shape. From minimum to maximum window shape, the sidelobe level increases and the gate resolution decreases. The choice of the window shape is always a trade-off between the gate resolution and the level of spurious sidelobes. The parameters of different window shapes are represented in the table below.

Window Shape	Bandpass Sidelobe Level	Gate Resolution (Minimum Gate Span)
Minimum	– 48 dB	$\frac{2.8}{F_{max}-F_{min}}$
Normal	– 68 dB	$\frac{5.6}{F_{max}-F_{min}}$
Wide	– 57 dB	$\frac{8.8}{F_{max}-F_{min}}$
Maximum	– 70 dB	$\frac{25.4}{F_{max}-F_{min}}$

DUT Low Pass Settings

The Time Domain Gating function has a setting to distinguish between the frequency lowpass DUT and the frequency bandpass DUT. The lowpass DUTs can operate with DC current such as cables or lowpass filters. The bandpass DUTs cannot operate with DC current such as band pass filters or high pass filers.

When the DUT Low Pass setting is OFF:

- The gating function makes no assumption about the DUT response at DC.
- The frequency settings can be arbitrary.

When the DUT Low Pass setting is ON:

- The value of the DUT response at DC is required to be known to the gating function.
- The frequency settings are required to be a harmonic frequency grid, where the frequency value at each frequency point is an integer multiple of the start frequency.

The DC value cannot be measured directly by the Analyzer. The Analyzer offers two options: the DC value is automatically extrapolated or manually set. The last option is used when the DUT response at DC is well known, for example, for a low loss cable the DC value is:

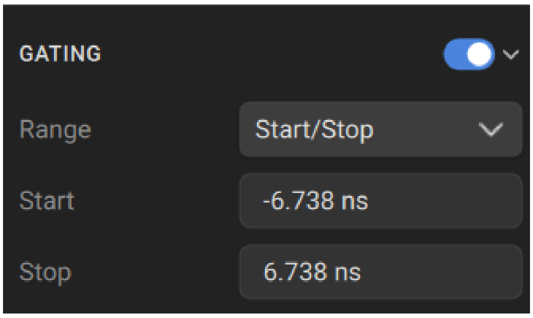
- "1" for open-ended cable.
- "-1" for a short-circuited cable.

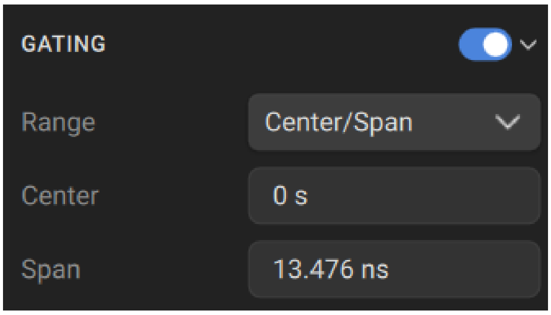
- "0" for a cable terminated with a matched load.

The Analyzer can set the harmonic frequency grid from the current frequency settings with one click.

NOTE	The following settings of the Gating function: DUT Low Pass ON/OFF, Set Frequency Low Pass, Extrapolate DC and DC value also set the corresponding settings of the Time Domain function (See Time Domain Transformation).

Time Domain Gate Activation





To enable/disable the time domain gating function:

Time Domain > Gating

Then use the toggle to turn it ON/OFF.

CALC:FILT:TIME:STAT	Turns the gating function ON/OFF.

NOTE	Time domain gating function is accessible only in linear frequency sweep mode.

Time Domain Gate Span

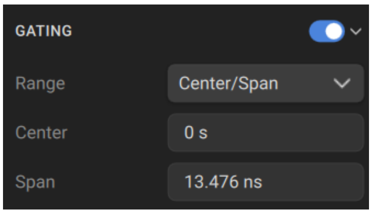
To set the start and stop of the time domain gate:

Time Domain > Gating > Start

Time Domain > Gating > Stop

CALC:FILT:TIME:STAR — Sets or reads out the gate start value of the gating function.

CALC:FILT:TIME:STOP — Sets or reads out the gate stop value of the gating function.



To set the center and span of the time domain gate:

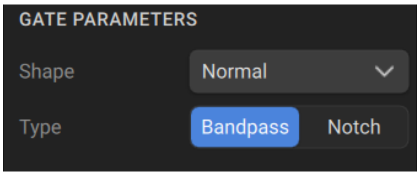
Time Domain > Gating > Center

Time Domain > Gating > Span

CALC:FILT:TIME:CENT — Sets or reads out the gate center value of the gating function.

CALC:FILT:TIME:SPAN — Sets or reads out the gate span value of the gating function.

Time Domain Gate Type



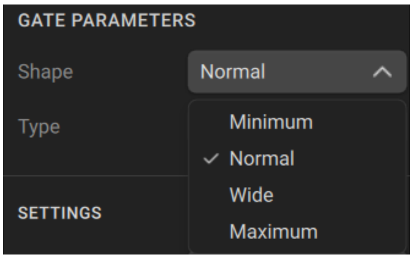
To select the gate type:

Time Domain > Gate Parameters

Toggle the type between **Bandpass** and **Notch**.

CALC:FILT:TIME — Sets or reads out the gate type of the gating function.

Time Domain Gate Shape Setting

	To set the time domain gate shape: **Time Domain > Gate Parameters > Shape > [Minimum | Normal | Wide | Maximum]**
CALC:FILT:TIME:SHAP	Sets or reads out the gate shape of the gating function.

Limit Test

The limit test is a function of automatic pass/fail judgment for the trace of the measurement result. The judgment is based on the comparison of the trace to the limit line set by the user.

The limit line can consist of one or several segments. Each segment checks the measured value for failure, whether it is an upper or lower limit. The limit line segment is defined by specifying the coordinates of the beginning (X0, Y0) and the end (X1, Y1) of the segment, and the type of the limit. The MAX or MIN limit types check if the trace falls outside of the upper or lower limit respectively.

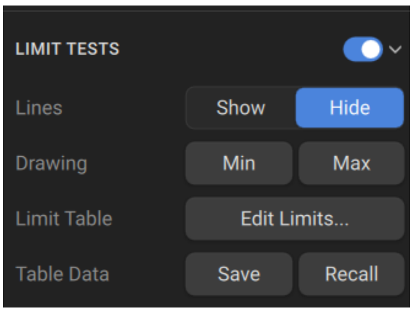
The limit line is set by the user in the limit table. Each row in the table describes one segment of the line. Limit table editing is described below. The table can be saved into a *.LIM file.

The display of the limit lines on the screen can be turned ON/OFF independently of the status of the limit test function.

The result of the limit test is indicated in the upper right corner of the diagram:

- If the measurement result passed the limit test, the trace number and the result will be seen: **Tr1: Limit Pass** (See figure above).
- If the measurement result failed, the result will be indicated in the following ways (See figure below):
  1. **Tr1:Limit Fail** will be displayed in upper right corner of the diagram.
  2. Fail sign will be displayed in red in the center of the window.
  3. The points of the trace, which failed the test will be highlighted in red.

Limit Test Enabling/Disabling



To enable/disable limit test function, use the following softkeys:

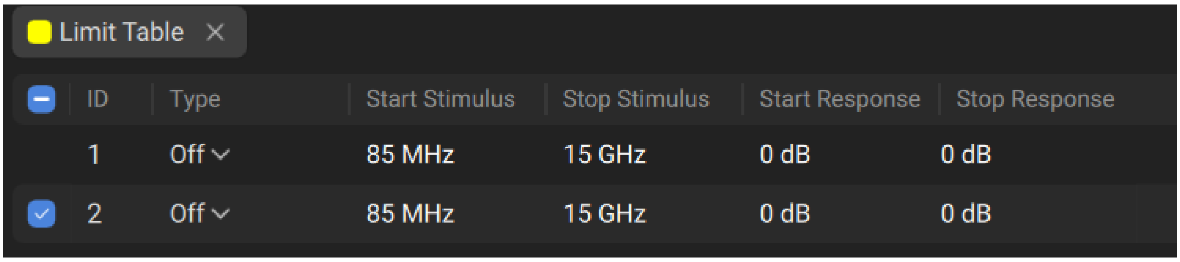
Limits > Limit Tests

Then use the toggle to turn it ON/OFF.

CALC:LIM	Turns the limit test ON/OFF.

Limit Line Editing

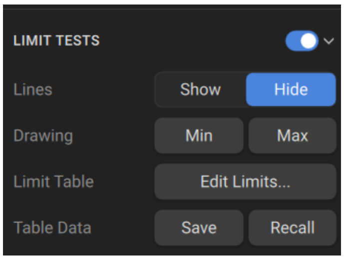
In the editing mode the limit table will appear in the lower part of the screen (See figure below). The limit table will be hidden when quitting the submenu.



Limit line table

Navigating within the table to enter the values of the following parameters of a limit test segment:

Type	Select the segment type among the following: • **MAX** — upper limit. • **MIN** — lower limit. • **SINGLE** — upper and lower limits in one frequency point. • **OFF** — segment not used for the limit test.
Star Stimulus	Stimulus value in the beginning point of the segment.
Stop Stimulus	Stimulus value in the ending point of the segment.
Start Response	Response value in the beginning point of the segment.
Stop Response	Response value in the ending point of the segment.



To enable/disable limit line:

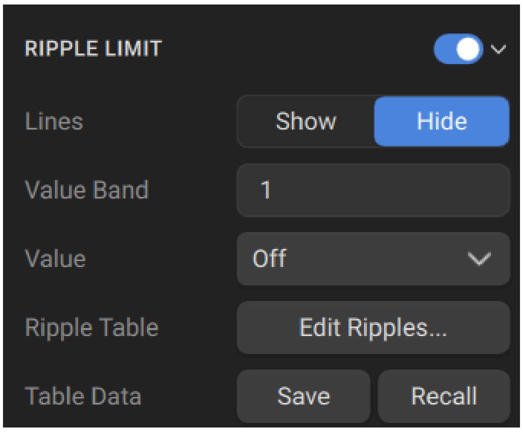
Limits > Lines [Show | Hide]

	To access the limit line editing mode: **Limits > Limit Table > Edit Limits...**
	These buttons appear when editing the limit table in the top right of the limit table under the graph. To add a new row in the table, click +. The new row will appear below the highlighted one. To delete a row from the table, click **-**. The highlighted row will be deleted.
CALC:LIM:DATA	Sets the data array, which is the limit line in the limit test function.
To save the table into *.LIM file, use: **Limits > Table Data [Save]**	
MMEM:STOR:LIM	Saves the ripple limit table into a file.
To open the table from a *.LIM file, use: **Limits > Table Data [Recall]**	
MMEM:LOAD:LIM	Recalls the limit table file. The file must be saved using the MMEM:STOR:LIM command.

Ripple Limit Test

The ripple limit test is an automatic pass/fail check of the measured trace data. The trace is checked against the maximum ripple value (ripple limit). The ripple value is the difference between the maximum and minimum response of the trace in the trace frequency band.

The ripple limit can include one or more segments. Each segment provides the ripple limit for the specific frequency band. A segment is set by the frequency band and the ripple limit value.



Ripple limits

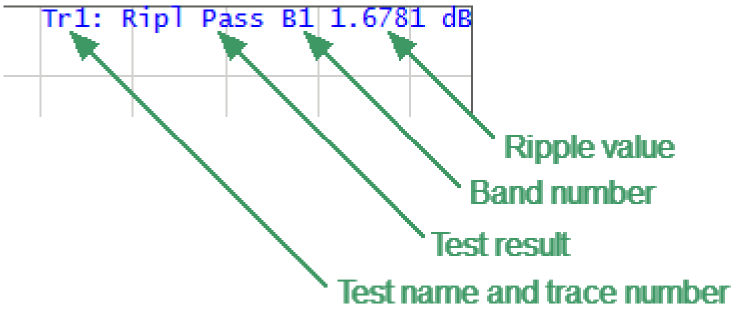
The ripple limit settings are set in the ripple limit table. Each row of the table describes the frequency band of the ripple limit value. The process of ripple limit table editing is described below. The table can be saved into a *.RML file.

The display of the limit lines on the screen can be turned ON/OFF independently of the status of the ripple limit test function.

The result of the ripple limit test is indicated in the upper right corner of the diagram:

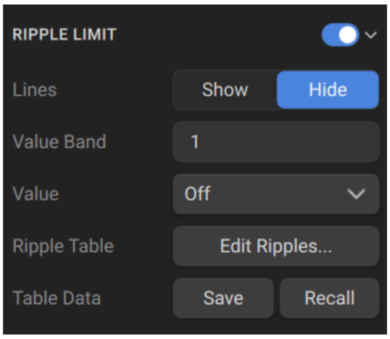
- If the measurement result passed the limit test, the trace number and the result will be seen: **Tr1: Ripl Pass**.
- If the measurement result failed, the result will be indicated in the following ways (See figure below):
  1. **Tr1: Ripl1: Fail** will be displayed in upper right corner of the diagram.
  2. Fail sign will be displayed in red in the center of the window.

The display of the ripple value can be enabled/disabled in the ripple limit test status line in the upper right corner of the diagram (See figure below). The ripple value is displayed for the band selected by the user. The ripple value can be represented as an absolute value or as a margin to the limit.



Ripple limit test status line

Ripple Limit Enabling/Disabling



To enable/disable ripple limit test function, use the following softkeys:

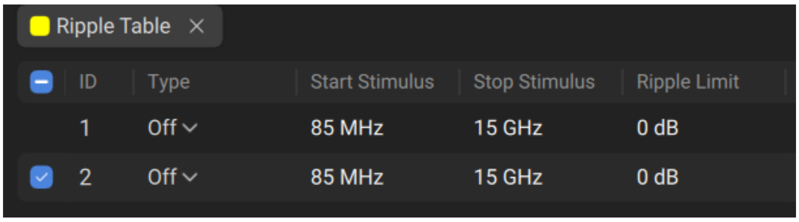
Limits > Ripple Limit

Then use the toggle to turn it ON/OFF.

CALC:RLIM	Turns the ripple limit test ON/OFF.

Ripple Limit Editing

In the editing mode, the limit table will appear in the lower part of the screen (See figure below). The limit table will be hidden when exiting the submenu.



Ripple limit table

Navigating within the table to enter the values of the following parameters of a ripple limit test segment:

Type	Select the segment type among the following: • **ON** — band used for the ripple limit test. • **OFF** — band not used for the limit test.
Start Stimulus	Stimulus value in the beginning point of the segment.
Stop Stimulus	Stimulus value in the ending point of the segment.
Ripple Limit	Ripple limit value.

To access the ripple limit editing mode:

Limits > Ripple Limit > Ripple Table > Edit Ripples...

These buttons appear when editing the ripple table in the top right of the ripple table under the graph.

To add a new row in the table, click **+**. The new row will appear below the highlighted one.

To delete a row from the table, click **-**. The highlighted row will be deleted.

CALC:RLIM:DATA	Sets the data array, which is the limit line for the ripple limit function.

To save the table into *.RML file, use:

Limits > Ripple Limit > Table Data [Save]

MMEM:STOR:RLIM	Saves the limit table into a file.

To open the table from a *.RML file, use:

Limits > Ripple Limit > Table Data [Recall]

MMEM:LOAD:RLIM	Recalls the limit table file.

To enter the number of the band, whose ripple value should be displayed, use the following softkeys:

Limits > Ripple Limit > Value Band

CALC:RLIM:DISP:SEL	Sets or reads out the number of the ripple limit test band selected for the ripple value display.

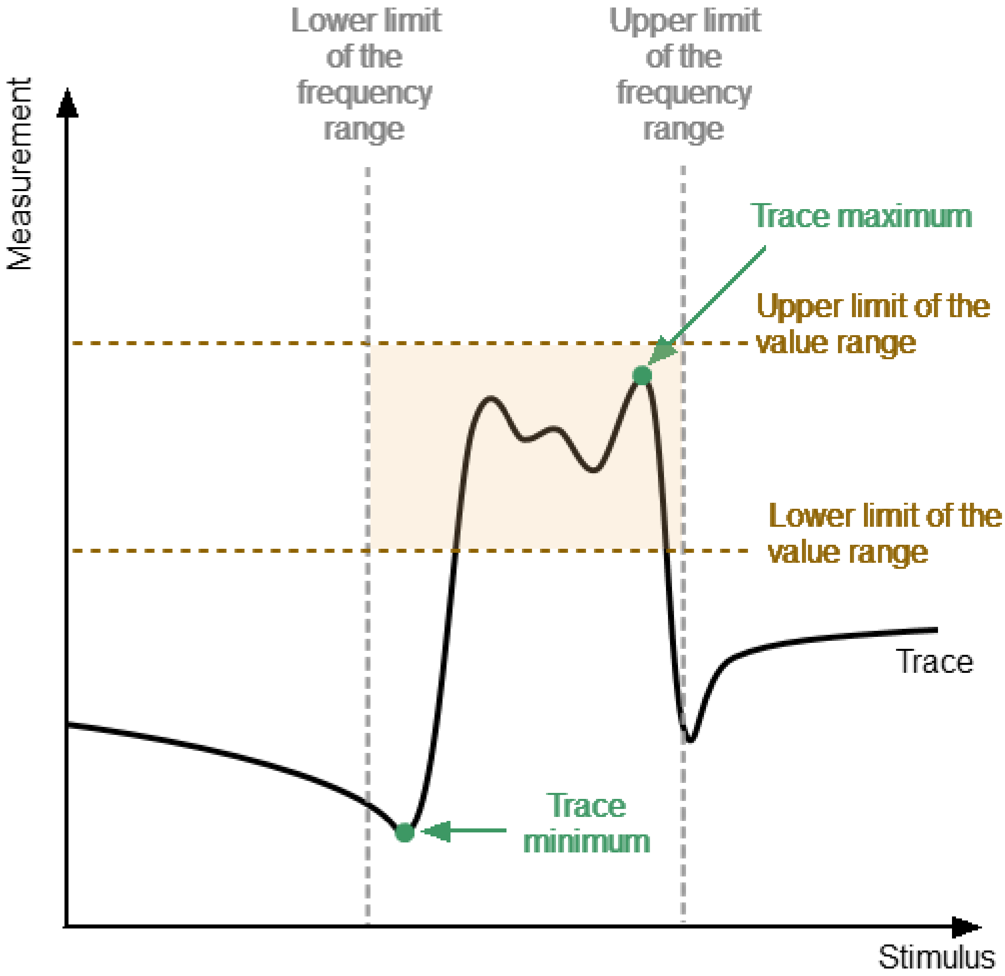
To enable/disable display of the ripple value:

Limits > Ripple Limit > Value [OFF | Absolute | Margin]

CALC:RLIM:DISP:VAL	Sets or reads out the number of the ripple limit test band selected for the ripple value display.

Peak Limits Test

The peak limits test function checks whether the trace point with the minimum (or maximum) value of the measured value falls within the specified limits of the frequency range and/or value range (see figure below). If the trace point minimum (or maximum) falls within the specified limits, the test is passed (test result "pass"). Otherwise, the test is failed (test result "fail").



Peak limits test

Peak Limits Test parameter

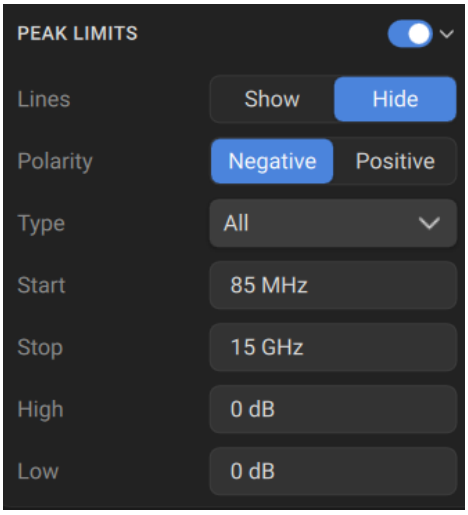
A description of the parameters of the peak limits test function in the software is shown in the table below.

Parameter	Definition
Limit Type	Selects the type of limit (one of the limits or both at the same time). Possible values: • **Stimulus** — checks whether the maximum (or minimum) point of the trace is within the specified frequency bandwidth. • **Response** — checks whether the maximum (or minimum) point of the trace falls within the value range. • **All** — checks the maximum (or minimum) trace point simultaneously within the frequency band and within the value range of the measured value.
Begin Stimulus	Lower bandwidth limit.
End Stimulus	Upper bandwidth limit.
Begin Response	Lower limit of the value range.
End Response	Upper limit of the value range.
Peak Polarity	Selects a trace point for inspection: • **Positive** — trace maximum. • **Negative** — trace minimum.

NOTE

The peak limit test for the **Stimulus** limit type can only be performed in the frequency domain.

Peak Limits Enabling/Disabling



To enable/disable peak limits test function:

Limits > Peak Limits

Then use the toggle to turn it ON/OFF.

Editing Search Parameters for Peak Limits

To select the limit type:

Limits > Peak Limits > Type

Then select the required type:

- **All**
- **Stimulus**
- **Response**

To enter the lower bandwidth limit, use **Limits > Peak Limits > Type: All or Type: Stimulus > Start**. To enter the upper bandwidth limit, use **Limits > Peak Limits > Type: All or Type: Stimulus > Stop**.

To enter the lower limit of the value range, use **Limits > Peak Limits > Type: All or Type: Response > Low**. To enter the upper limit of the value range, use **Limits > Peak Limits > Type: All or Type: Response > High**.

To select a trace point to check, use **Polarity [Negative | Positive]**.

Peak Limits Test Display Management

The display of the peak limits on the screen can be turned on/off.

The result of the peak limits test is indicated in the upper right corner of the diagram:

- If the measurement result passed the peak limit test, the trace number and the result will be seen: **Tr1: PkLim Pass**.
- If the measurement result failed, the result will be indicated in the following ways:
  1. **Tr1: PkLim Fail** will be displayed in upper right corner of the diagram.
  2. Fail sign will be displayed in red in the center of the window.

State Saving and Data Output

The following section describes the processes of saving and recalling:

- The set parameters of the Analyzer, calibration, measured, and memorized data are stored in the Analyzer status file and can be loaded repeatedly (See Analyzer States).
- The states of the channels are stored into the Analyzer's inner memory. Up to 4 states can be stored while the Analyzer is running. When the Analyzer is powered off, the contents of the state memory are destroyed.
- Trace data in a *.CSV file (See Trace Data CSV Files).
- DUT S-parameters in a Touchstone file (See Trace Data Touchstone Files).

Analyzer State

The Analyzer state, calibration and measured data can be saved on the hard disk to an Analyzer state file and later uploaded back into the Analyzer software. The following four types of saving are available:

State	The Analyzer settings.
State & Cal	The Analyzer settings and the table of calibration coefficients.
State & Trace	The Analyzer settings and data traces[1].
All	The Analyzer settings, table of calibration coefficients, and data traces and memory[1].
State & Cal & Mem	The Analyzer settings, table of calibration coefficients and memory.
1 When recalling the state with saved data traces, the trigger mode will be automatically set to «Hold» so that the recalled traces are not erased by currently measured data.	

The Analyzer settings that are saved into the Analyzer state file are parameters that can be set in the following sub-levels of the softkey bar:

- All the parameters in the **Stimulus**
- All the parameters in the **Measurement**
- All the parameters in the **Format**
- All the parameters in the **Scale**
- All the parameters in the **Average**
- All the parameters in the **Display** except for **Properties**
- All the parameters of the **Markers**
- All the parameters of the **Analysis**
- **Ref Source** and **System Correction** parameters in the **System**

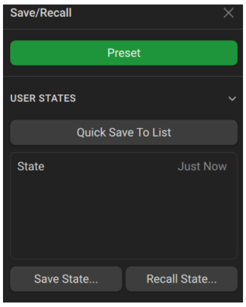
To save and recall a state file, ten softkeys labeled **State01, ... State10** can be used. Each of the softkeys correspond to a *.STA file with the same name.

To have the Analyzer state automatically recalled after each start of the instrument use the *Autorecall.sta* file. Use the **Autorecall** softkey to save the corresponding file and thus enable this function.

To disable the automatic recall of the Analyzer state, delete the *Autorecall.sta* file using the specific softkey.

The files can be saved and recalled with arbitrary names. To save, use the **File...** softkey, which will open the **Save as** dialog box.

Analyzer State Saving

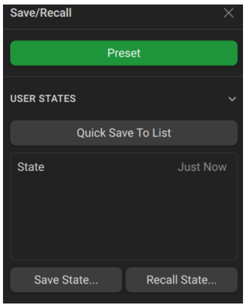


To save the state, use the following softkeys:

Save/Recall > Save State

MMEM:STOR	Saves the Analyzer state into a file.

Analyzer State Recalling



To recall the state from an Analyzer state file:

Save/Recall > Recall State

MMEM:LOAD	Recalls the specified Analyzer state file.

Trace Data CSV File

Trace data can be saved as a *.CSV file (comma separated values). The *.CSV file contains comment and trace data lines. Comments start from the «!» symbol.

Before saving the *.CSV file, set the trace type, value delimiter type, and other parameters in the **Save Trace Data** submenu (See the table below). Then, click the **Save...** button to save the values to the file.

Parameter	Definition
Scope	Type of trace to be saved: • **Active Trace**. • **All Traces of Chan** — all traces of the active channel.
Format	Data save format: • **Displayed** — the format in which the trace is set (See Format Setting). • **Real-Imag** — real and imaginary parts. • **db-Angle** — logarithmic magnitude in dB and phase in degrees.
Comment	Enable/disable the entry in the comment file. The comment contains 3 lines: 1. Model, serial number, software version. 2. Save date (in the dd.mm.yyyy hh:mm:ss format). 3. The name of the saved parameters and their dimensionality.
Stimulus	Enable/disable recording to the file frequency at measurement point.
Decimal Separator	The type of delimiters between stored values, as well as the type of decimal separator: • **Local** — delimiters defined in regional settings are used. • **Point** — decimal separator is point, value separator is comma.

The trace data is saved to *.CSV in the following format:

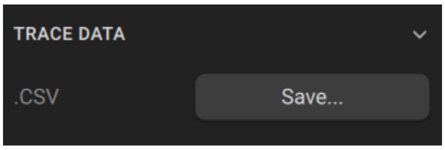
! Comment		
F[0],	Data1,	Data2
F[1],	Data1,	Data2
. . .		
F[N],	Data1,	Data2

F[n] — frequency at measurement point n.

Data1 — trace response in rectangular format, real part in Smith chart and polar format.

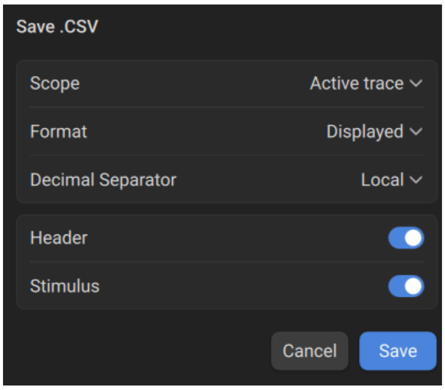
Data2 — zero in rectangular format, imaginary part in Smith chart and polar format.

Editing saving parameters



To open save trace submenu , use:

Save/Recall > Trace Data > Save...



To select the type of trace, use the **Scope** dropdown.

Then select the required type:

- **Active Trace**
- **All Traces of Channel**

MMEM:STOR:FDAT:SCOP	Sets whether the active trace or all active traces will be saved using the MMEM:STOR:FDAT command.

	To select the format for saving data, use the **Format** dropdown. Then select the required format: • **Displayed** • **Real-Imag** • **db-Angle**
MMEM:STOR:FDAT:FORM	Sets the data format when the *.CSV file is saved using the MMEM:STOR:FDAT command.
	To enable/disable writing to a file frequency at measurement point, use the **Stimulus** toggle.
MMEM:STOR:FDAT:STIM	Turns the column with the stimulus data in the *.CSV file saved with the MMEM:STOR:FDAT command ON/OFF.
	To select the type of separators, use the **Decimal Separator** dropdown. Then select the required format: • **Local** • **Point**
MMEM:STOR:FDAT:SEP	Sets the separators used when the *.CSV file is saved with the MMEM:STOR:FDAT command.

CSV File Saving

To save the .CSV file, use the Save button at the bottom right of the Trace Data .CSV Save popup.

MMEM:STOR:FDAT	Saves the data of one or several traces to a CSV file.

Trace Data Touchstone File

The Analyzer allows to save S-parameters to a Touchstone file. Files in this format are typical for most circuit simulator programs. The Touchstone file contains frequency values and S-parameters.

The Touchstone file saving function is applied to individual channels. Activate the channel to use this function (See Selection of Active Trace/Channel).

The *.S1P files are used for saving S11 and S22 parameters of a one-port device.

The *.S2P files are used for saving all four S-parameters of a two-port device.

NOTE	The Analyzer allows the measurement of only S11 and S21 parameters. When *.S2P is saved, the missing S-parameters S12 and S22 are filled in as zeroes.

The Touchstone file contains comments, header, and trace data lines. The header starts from the «#» symbol. Comments start from the «!» symbol. Comment contains following strings:

- Model, serial number, software version.
- Save date (in dd.mm.yyyy hh:mm:ss format).
- The name of the saved parameters and their units.

The *.S1P Touchstone file for one-port measurements:

```
! Comments

# Hz S FMT R Z0

F[0]      {S11}'    {S11}"

F[1]      {S11}'    {S11}"

          . . .

F[N]      {S11}'    {S11}"
```

The *.S2P Touchstone file for two-port measurements:

```
! Comments

# Hz S FMT R Z0

F[0]    {S11}'  {S11}"  {S21}'  {S21}"  {S12}'  {S12}"  {S22}'  {S22}"

F[1]    {S11}'  {S11}"  {S21}'  {S21}"  {S12}'  {S12}"  {S22}'  {S22}"

. . .

F[N]    {S11}'  {S11}"  {S21}'  {S21}"  {S12}'  {S12}"  {S22}'  {S22}"
```

Hz — frequency measurement units (**kHz, MHz, GHz**);

FMT — data format:

- **RI** — real and imaginary parts;
- **MA** — linear magnitude and phase in degrees;
- **DB** — logarithmic magnitude in dB and phase in degrees;

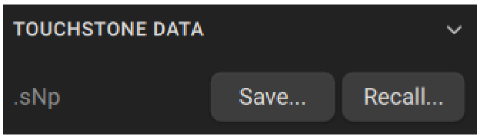
Z0 — reference impedance value;

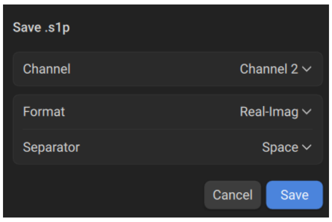
F[n] — frequency at measurement point n;

{…}' — {real part (RI) | linear magnitude (MA) | logarithmic magnitude (DB)};

{…}" — {imaginary part (RI) | phase in degrees (MA) | phase in degrees (DB)}.

Touchstone File Saving





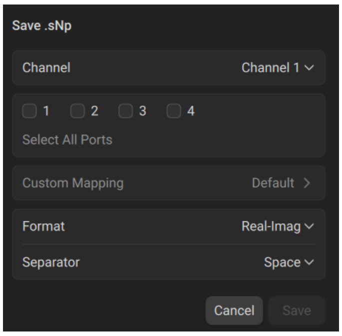
To select the saving type, use the following softkeys:

Save/Recall > Save Data To Touchstone File > Type

Then select the required Touchstone type:

- **1-Port (s1p)**
- **2-Port (s2p)**

MMEM:STOR:SNP:TYPE:S1P	Sets and reads out the one-port Touchstone file type (*.S1P) and the port number.
MMEM:STOR:SNP:TYPE:S2P	Sets and reads out the two-port Touchstone file type (*.S2P) and the ports number.
MMEM:STOR:SNP:TYPE?	Reads out the type of Touchstone file.



To select the port numbers for one/two/three-port Touchstone file, use the following softkeys:

Save/Recall > Touchstone Data

Then select the required port numbers:

- **Select Port (s1p) > [1 | 2 | 3 | 4]**
- **Select Ports (s2p) > [1-2 | 1-3 |1-4 | 2-3 | 2-4 | 3-4]**
- **Select Ports (s3p) > [1-2-3 | 1-2-4 |1-3-4 | 2-3-4]**

To select the data format:

Save/Recall > Touchstone Data Save > Format

Then select the required Touchstone format:

- **Real-Imaginary**
- **Magnitude-Angle**
- **dB-Angle**

MMEM:STOR:SNP:FORM	Sets and reads out the data format for the S-parameter.

To select the Touchstone separator type, use the following softkeys:

Save/Recall > Touchstone Data Save > Separator

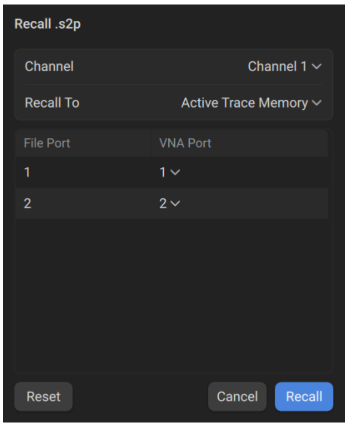
Then select the required Touchstone separator:

- **Tab**
- **Space**

MMEM:STOR:SNP:SEP	Sets and reads out the Touchstone file separator symbol.

Touchstone File Recalling

The Analyzer allows to recall data from the Touchstone files. Data can be loaded to memory traces or to data traces. When loading data to data traces, the Analyzer switches to hold mode to avoid writing over the recalled data with current data. When loading data to the memory traces, the sweep hold does not occur.



To load the data from the Touchstone file, use one of the softkeys:

Save/Recall >Touchstone Data Recall

Then select the required data loading method:

- **Active Trace Memory** — loading data to the active trace memory.
- **All Traces Memory** — loading data to the memory of all traces.
- **Static Trace**.

To select the port mapping, use the VNA Port dropdown to assign the VNA port to the file port.

MMEM:LOAD:SNP	Loads the Touchstone file with the specified name to the measured S-parameters of the active channel.
MMEM:LOAD:SNP:TRAC:MEM	Loads the Touchstone file with the specified name to the memory trace.

System Settings

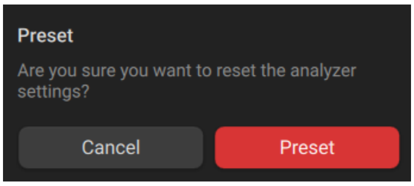
Analyzer Presetting

The Analyzer presetting feature allows to restore the default settings of the Analyzer.

The default settings of the Analyzer are specified in Default Settings Table.

To preset the Analyzer, use the Preset button in the Quick Access Toolbar.

If you have a previous file loaded or have altered settings, you will get the following popup:



SYST:PRES	Resets the Analyzer to factory settings.

Graph Printing

This section describes the print/save procedures for graph data. It can be found under Save/Recall > Report.

The print function is provided with the preview feature, which allows to view the image to be printed on the screen, and/or save it to a file.

The graphs can be printed using three different applications:

- MS Word (Windows only).
- Image Viewer for Windows (Windows only).
- Print Wizard of the Analyzer (Windows & Linux).

NOTE	The MS Word application must be installed on the Windows system.

NOTE	The Print Wizard requires at least one printer to be installed in Windows.

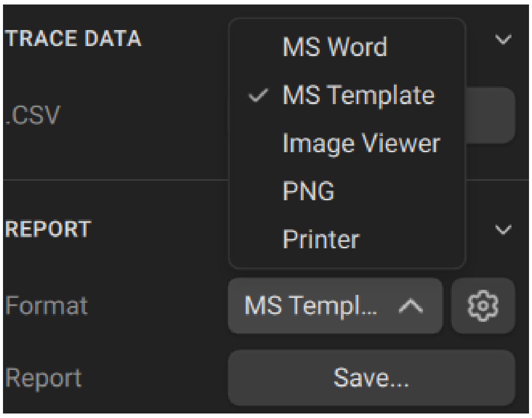
Print color can be selected before the image is transferred to the printing application:

- Color (no changes).
- Gray scale.
- Black & white.

The image can also be inverted before it is transferred to the printing application.

The current date and time can be added before the image is transferred to the printing application.

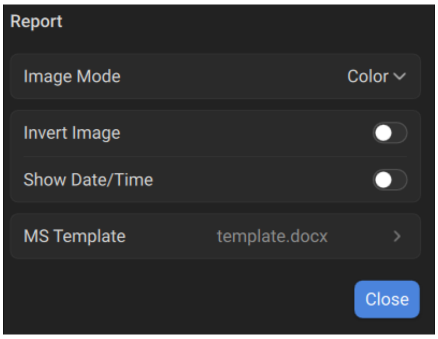
To open a print menu, use the ⚙ in Save/Recall > Report > Format



Then select the printing application use the dropdown by Format:

- **MS Word**
- **MS Template**
- **Image Viewer**
- **PNG**
- **Printer**

HCOP	Prints out the image displayed on the screen without previewing.
MMEM:STOR:IMAG	Saves the display image in BMP or PNG format into a file.



To set the print color, use **Image Mode** dropdown after clicking the ⚙.

Then select the required color, using one of the following softkeys:

- **Color**
- **Gray**
- **Black & White**

HCOP:PAIN	Sets or reads out the color chart for the image printout.

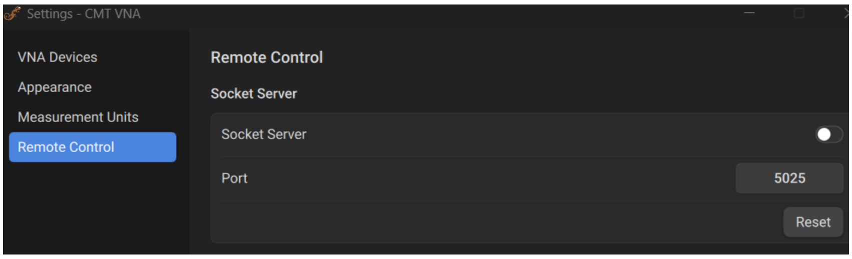
If necessary, invert the image by using the **Invert Image** toggle.

If necessary, select printing of date and time by using the **Show Date & Time** toggle.

HCOP:DATE:STAM	Turns the date and time printout in the upper right corner of the image ON/OFF.
HCOP:IMAG	Sets or reads out the color chart for the image printout.

Remote Control

Navigate to Settings > Remote Control to find the socket server settings.



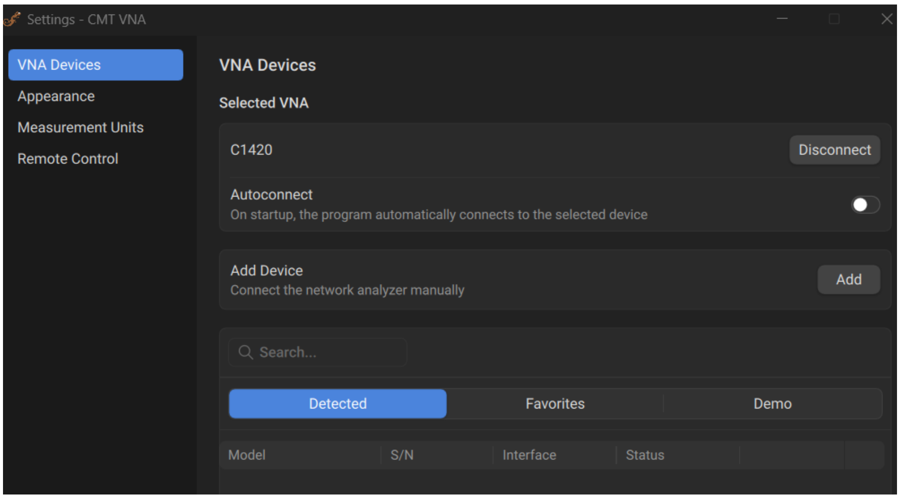
To turn on the Socket Server, use the toggle.

Set the Socket Server Port using the Port textbox and type the desired port.

Use the Reset button to reset the socket server settings.

VNA Devices

Navigate to Settings > VNA Devices to access the menu to change the connected device, add a device, or go to Demo mode.



The Select VNA section shows which VNA is currently connected to the program. Use the Disconnect button to disconnect it.

Toggle ON the Autoconnect feature if you would like this device to autoconnect to the program on startup. Toggle OFF to turn this feature off.

The add a device, or connect one manually, use:

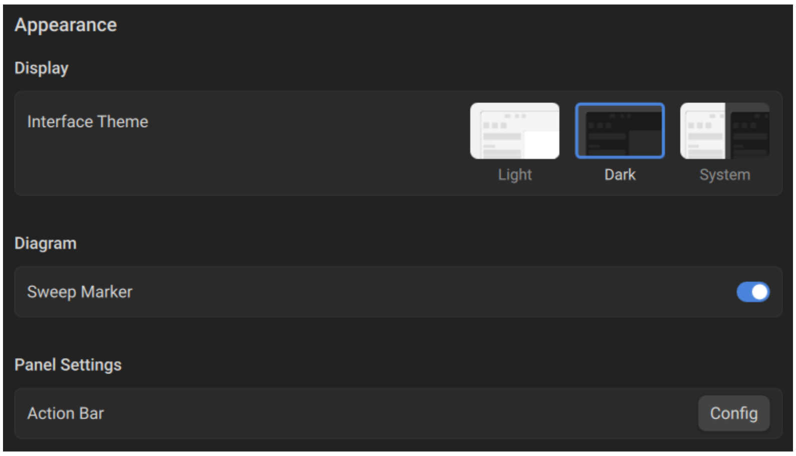
Settings > VNA Devices > Add Device

The table at the bottom of the popup shows which devices are currently connected. It gives the Model name, the serial number, the interface, and the status information.

To see information about Demo mode, see Demo Mode.

Appearance

This menu allows the user to set the appearance of CMT VNA.

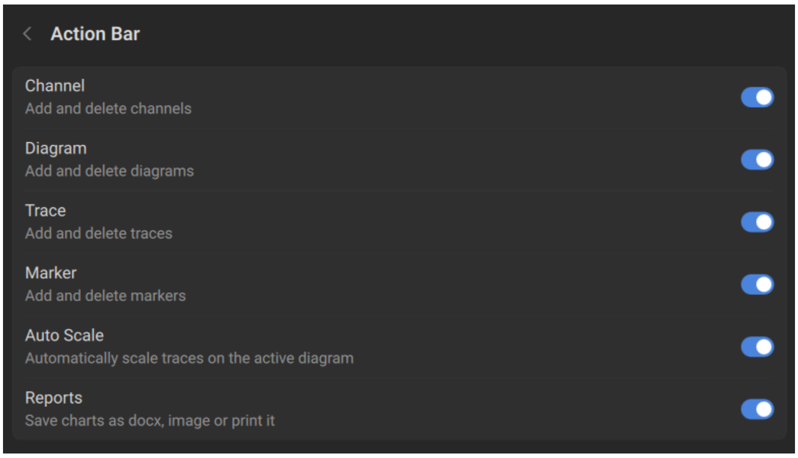


Use the Display section to select between Light mode, Dark mode, and System match mode.

The toggle ON/OFF sweep markers on the diagram:

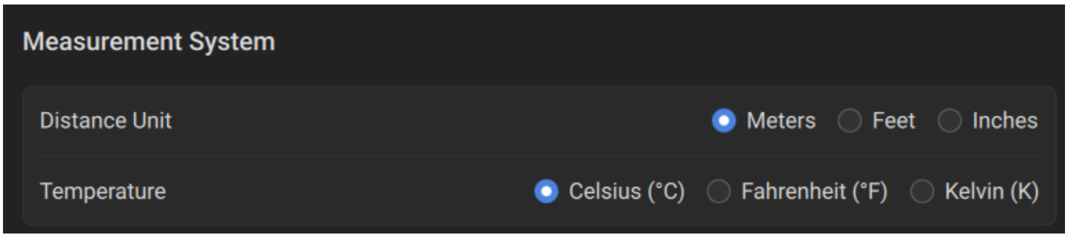
Settings > Appearance > Sweep Marker [ON|OFF]

The Panel Settings sections allows the user to configure which buttons are visible on the Quick Action Toolbar. Use the toggles to add or remove each button from the bar.

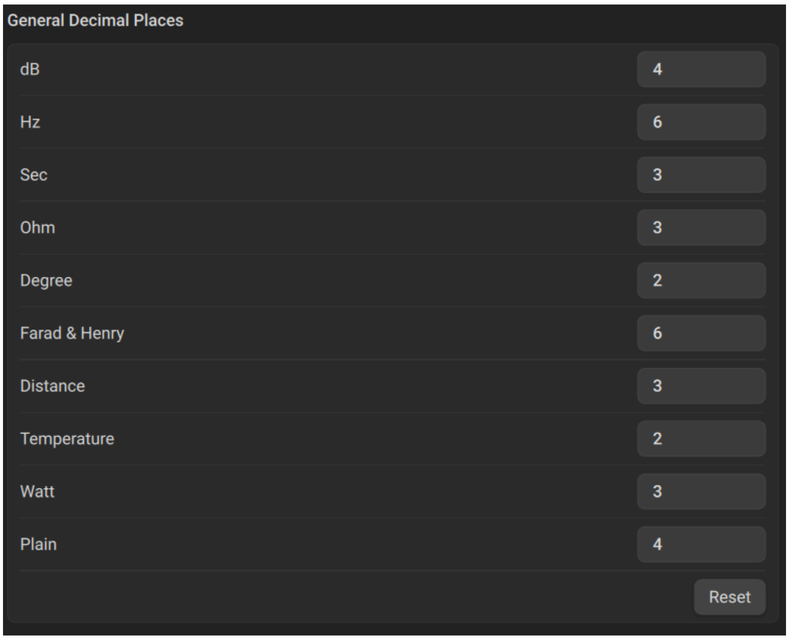


Measurement Units

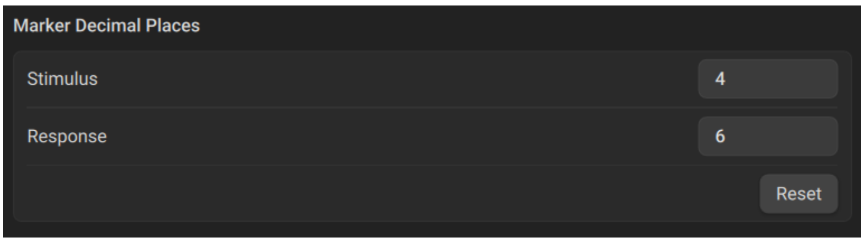
This section of the Settings allows the user to set the units for a given measurement in the software.



Set the overall distance units and the temperature units under **Settings > Measurement Units > Measurement System**.



Set the number of decimal places for the following units under **Settings > Measurement Units > General Decimal Places:** dB, Hz, Sec, Ohm, Degree, Farad & Henry, Distance, Temperature, Watt, and Plain (all numbers other that those with the above-listed units).

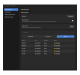


Set the number of decimal points for markers under **Settings > Measurement Units > Marker Decimal Places.** This can be set for both the stimulus and response markers.

Demo Mode

Demo mode is designed to simulate DUT measurement. The measurement results of the DUT are pre-recorded in the software memory. Any Analyzer model can be selected from the list of supported devices in demo mode.

NOTE The simulation of the Analyzer in demo mode may differ from the real measurements of the analyzer. For example, the accuracy of the sweep time dependence on the IF filter setting is not guaranteed.



To enable/disable the demo mode:

Settings > VNA Devices [Demo] > Connect/Disconnect

The Connect/Disconnect button can be seen by hovering over the right side of the device menu.

NOTE Demo mode is automatically enabled when the software is downloaded. Connect a device in the VNA Devices menu to disable demo mode

Maintenance and Storage

The following section describes the proper maintenance and storage procedures for the Analyzer.

Maintenance Procedures

This section describes the guidelines and procedures of maintenance, which will ensure fault-free operation of the Analyzer.

The maintenance of the Analyzer consists of cleaning the instrument, factory calibrations, and regular performance tests.

Instrument Cleaning

This section provides the cleaning instructions required for maintaining proper operation of the Analyzer.

To remove contamination from parts other than test ports or any connectors of the Analyzer, wipe them gently with a soft cloth that is dry or wetted with a small amount of water and wrung tightly.

It is essential to always keep the test ports clean, as any dust or stains on them can significantly affect the measurement capabilities of the instrument. To clean the test ports (as well as other connectors of the Analyzer), use the following procedure:

- Using compressed air, remove or loosen the contamination particles.
- Clean the connectors using a lint-free cleaning cloth wetted with a small amount of ethanol and isopropyl alcohol (when cleaning a female connector, avoid snagging the cloth on the center conductor contact fingers by using short strokes).
- Dry the connector with low-pressure compressed air.

Always completely dry a connector before using it.

Never use water or abrasives for cleaning any connectors on the Analyzer. Do not allow alcohol contact on the surface of the connector.

When connecting male-female coaxial connectors, always use a calibrated torque wrench.

WARNING	Never perform cleaning of the instrument if the power cable is connected to the power outlet. Never clean the internal components of the instrument.

Factory Calibration

Factory calibration is a regular calibration performed by the manufacturer or an authorized service center. It is recommended to send the analyzer for factory calibration every three years.

Performance Test

The performance test is done to verify that the performance of the Analyzer is up to the published specifications.

A performance test of the Analyzer should be performed in accordance with Performance Test Instructions.

The performance test period is one year.

Download VNA performance test from https://coppermountaintech.com/download-files/.

Storage Instructions

Before first use, store the Analyzer in the factory package at a temperature from -50 ºC to+70 ºC (-58 °F to 158 °F) and relative humidity up to 90% at 25 ºC (77 °F).

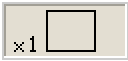
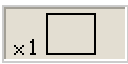
After the analyzer has been removed from the factory packaging and while being used, it should be stored at a temperature from+5 ºC to+40 ºC and relative humidity up to 90% at 25 ºC (1 °F to 104 °F).

Be sure to keep the storage facilities free from dust, acidic or alkaline fumes, volatile gases, and other chemicals, which can cause corrosion.

Annexes

Default Settings Table

Default values defined in the process of the initial factory setup.

Parameter Description	Default Setting	Parameter Setting Object
Data Saving Type	State and Calibration	Analyzer
Touchstone Data Format	Real-Imaginary	Analyzer
Allocation of Channels		Analyzer
Active Channel Number	1	Analyzer
Marker Value Identification Capacity (Stimulus)	7 digits	Analyzer
Marker Value Identification Capacity (Response)	4 digits	Analyzer
Marker Table	OFF	Analyzer
Reference Frequency Source	Internal	Analyzer
Trigger Signal Source	Internal	Analyzer
Reference Channel Error Correction	ON	Analyzer
System Correction	ON	Analyzer
Allocation of Traces		Channel
Vertical Divisions	10	Channel
Channel Title Bar	OFF	Channel

Parameter Description	Default Setting	Parameter Setting Object
Channel Title	Empty	Channel
«FAIL» Label Display (Limit Test)	OFF	Channel
Segment Sweep Frequency Axis Display	Frequency Order	Channel
Traces per Channel	1	Channel
Active Trace Number	1	Channel
Marker Coupling	ON	Channel
Sweep Type	Linear Frequency	Channel
Number of Points	201	Channel
Stimulus Start Frequency	Instrument min.	Channel
Stimulus Stop Frequency	Instrument max.	Channel
Stimulus CW Frequency	Instrument min.	Channel
Stimulus Start Power Level	Instrument min.	Channel
Stimulus Stop Power Level	Instrument max.	Channel
Stimulus Power Level	0 dBm	Channel
Stimulus Power Slope	0 dBm	Channel
Stimulus IF Bandwidth	10 kHz	Channel
Sweep Measurement Delay	0 sec.	Channel
Sweep Range Setting	Start / Stop	Channel

Parameter Description	Default Setting	Parameter Setting Object
Number of Segments	1	Channel
Points per Segment	2	Channel
Segment Start Frequency	Instrument min.	Channel
Segment Stop Frequency	Instrument min.	Channel
Segment Sweep Power Level	0 dBm	Channel
Segment Sweep IF Bandwidth	10 kHz	Channel
Segment Sweep Measurement Delay	0 sec.	Channel
Segment Sweep Power Level (Table Display)	OFF	Channel
Segment Sweep IF Bandwidth (Table Display)	OFF	Channel
Segment Sweep Measurement Delay (Table Display)	OFF	Channel
Segment Sweep Range Setting	Start / Stop	Channel
Averaging	OFF	Channel
Averaging Factor	10	Channel
Trigger Mode	Continuous	Channel
Table of Calibration Coefficients	Empty	Channel
Error Correction	OFF	Channel
Port Z Conversion	OFF	Channel

Parameter Description	Default Setting	Parameter Setting Object
Port 1 Simulated Impedance	Instrument Nominal	Channel
Port 2 Simulated Impedance	Instrument Nominal	Channel
Port 1 De-embedding	OFF	Channel
Port 2 De-embedding	OFF	Channel
Port 1 De-embedding S-parameter File	Empty	Channel
Port 2 De-embedding S-parameter File	Empty	Channel
Port 1 Embedding	OFF	Channel
Port 2 Embedding	OFF	Channel
Port 1 Embedding User File	Empty	Channel
Port 2 Embedding User File	Empty	Channel
Measurement Parameter	S11	Trace
Trace Scale	10 dB / Div.	Trace
Reference Level Value	0 dB	Trace
Reference Level Position	5 Div.	Trace
Data Math	OFF	Trace
Phase Offset	0°	Trace
Electrical Delay	0 sec.	Trace

Parameter Description	**Default Setting**	**Parameter Setting Object**
S-parameter Conversion	OFF	Trace
S-parameter Conversion Function	Z: Reflection	Trace
Trace Display Format	Logarithmic Magnitude (dB)	Trace
Time Domain Transformation	OFF	Trace
Time Domain Transformation Start	–10 nsec.	Trace
Time Domain Transformation Stop	10 nsec.	Trace
Time Domain Kaiser-Beta	6	Trace
Time Domain Transformation Type	Bandpass	Trace
Time Domain Gate	ON	Trace
Time Domain Gate Start	–10 ns	Trace
Time Domain Gate Stop	10 ns	Trace
Time Domain Gate Type	Bandpass	Trace
Time Domain Gate Shape	Normal	Trace
Smoothing	OFF	Trace
Smoothing Aperture	1%	Trace
Trace Display Mode	Data	Trace
Limit Test	OFF	Trace

Parameter Description	Default Setting	Parameter Setting Object
Limit Line Display	OFF	Trace
Defined Limit Lines	Empty	Trace
Number of Markers	0	Trace
Marker Position	Instrument min.	Trace
Marker Search	Maximum	Trace
Marker Tracking	OFF	Trace
Marker Search Target	0 dB	Trace
Marker Search Target Transition	Both	Trace
Marker Search Peak Polarity	Positive	Trace
Marker Search Peak Excursion	3 dB	Trace
Bandwidth Parameter Search	OFF	Trace
Marker Search Bandwidth Value	–3 dB	Trace
Marker Search Range	OFF	Trace
Marker Search Start	0	Trace
Marker Search Stop	0	Trace

ACM Operating manual

This Operating Manual contains information on design, specifications, functional overview, and detailed operation procedures of the Copper Mountain Technologies Automatic Calibration Modules (hereinafter referred to as Modules). Use the navigation tools on the left of the window to access the sections.

General Overview

The Module is designed for calibration (error correction) of Vector Network Analyzers in automatic mode.

Calibration is performed by automatically connecting the reflection and transmission impedance states to the VNA test ports.

Calibration determines systematic errors in accordance with the VNA model. The process of mathematical compensation (numerical reduction) for measurement systematic errors is called error correction.

Using the Module instead of a mechanical calibration kit has several advantages, which ensure high measurement accuracy and a longer service life of the VNA test ports. The measurement accuracy is achieved using precision Module standards (states) descriptions, by the stability of the selected configuration, and by the application of temperature drift functions and self-diagnosis in the form of confidence check. Single module connection during calibration allows to:

- Extend the VNA ports service life.
- Reduce technical staff workload and risk of human error.
- Make the measurement process most efficient.

The Module control protocol is based on the USBTMC-USB488 standard.

Modification

The Module differ in operating frequency range and in the number of ports. Their functional features are briefly described in the table below.

During calibration, the Modules are controlled by the VNA software installed on the connected PC. The USB 2.0 interface is used for control.

The Modules feature several hardware configurations depending on the connector types of PORT A, PORT B and, if available, PORT C and PORT D. To view the possible connector type front and side views for each Module, click on the name of the desired Module in the table below.

The Module delivery package is specified in Delivery Kit.

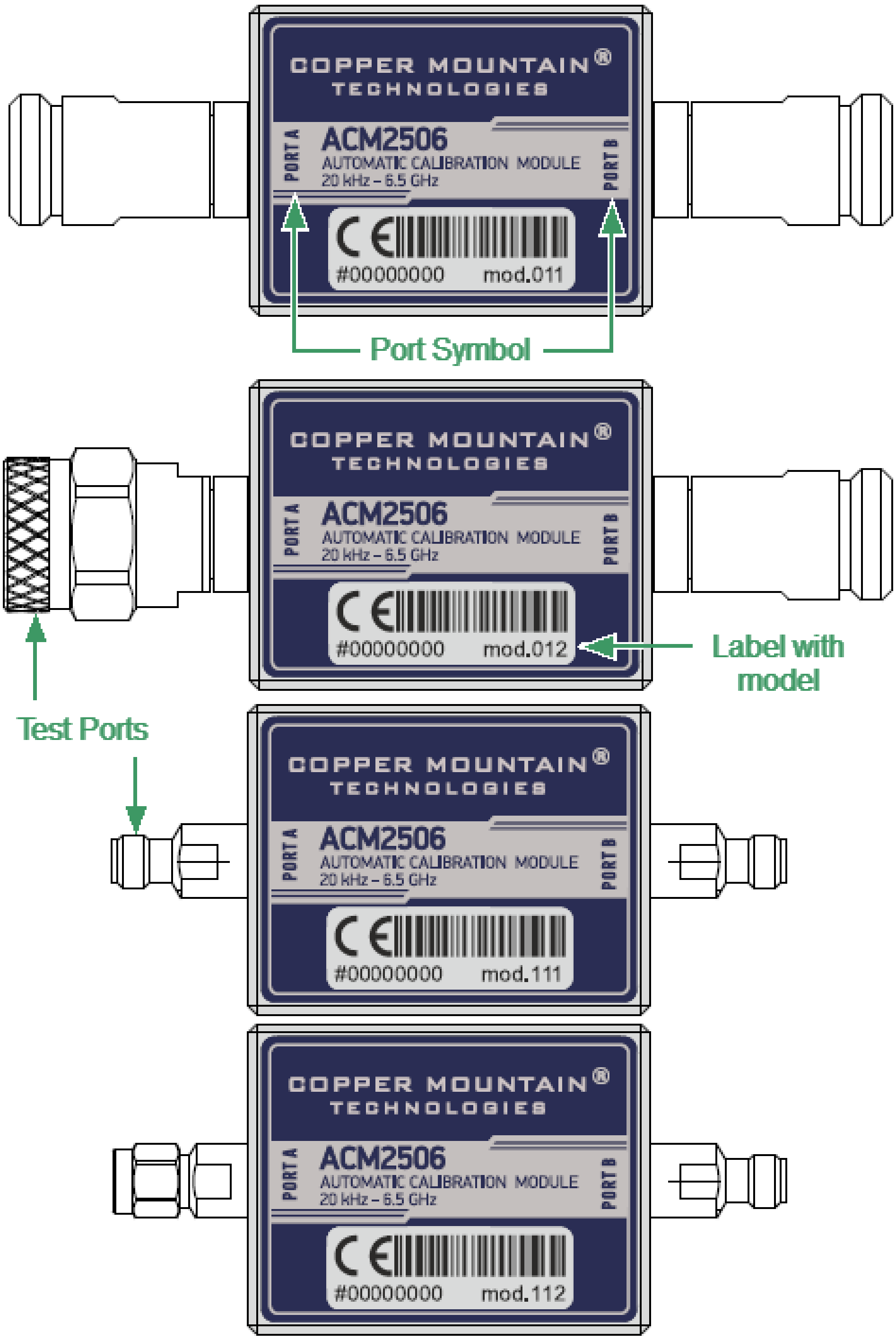
Functional Features

<table>
<tr><th>Module</th><th>Frequency range</th><th>Supported calibrations</th><th>Features</th></tr>
<tr><td colspan="4">50 Ohm two-port Modules</td></tr>
<tr><td>ACM2506</td><td>20 kHz to 6.5 GHz</td><td rowspan="6">Full one-port
One-path two-port</td><td rowspan="6">Unknown thru
Thermal compensation
User characterization
Automatic orientation
Confidence check</td></tr>
<tr><td>ACM2509</td><td>20 kHz to 9 GHz</td></tr>
<tr><td>ACM2520</td><td>100 kHz to 20 GHz</td></tr>
<tr><td>ACM2543</td><td>10 MHz to 44 GHz</td></tr>
<tr><td>ACM6000T</td><td>20 kHz to 6 GHz</td></tr>
<tr><td>ACM8000T</td><td>100 kHz to 8 GHz</td></tr>
<tr><td colspan="4">75 Ohm two-port Modules</td></tr>
<tr><td>ACM2708</td><td>20 kHz to 8 GHz</td><td>Full one-port
One-path two-port</td><td>Unknown thru
Thermal compensation</td></tr>
</table>

Module	Frequency range	Supported calibrations	Features
ACM4000T	20 kHz to 4 GHz		User characterizatio n Automatic orientation Confidence check
50 Ohm four-port Modules			
ACM4509	100 kHz to 9 GHz	Full one-port One-path two-port	Unknown thru Thermal compensation User characterizatio n Automatic orientation Confidence check
ACM4520	100 kHz to 20 GHz		
ACM8400T	100 kHz to 8 GHz		
1 The upper frequency point of ACM2520 and ACM4520 with type N connectors is 18 GHz. 2 The upper frequency point of ACM2543 with 2.92 mm connectors is 40 GHz.			

ACM2506

The front panels of the different models of ACM2506 are shown in the figure below.



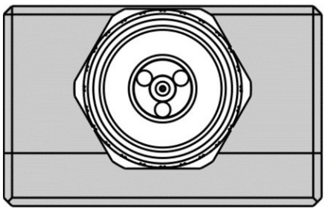
Front panel ACM2506

Parts of the ACM2509

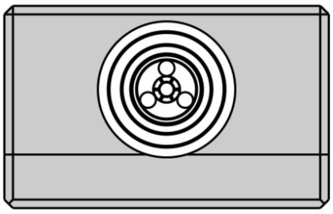
Test Port

The test ports are designed for connection to VNA being calibrated. The VNA connectors, the cross sections of which were calibrated, are referred to as its test ports.

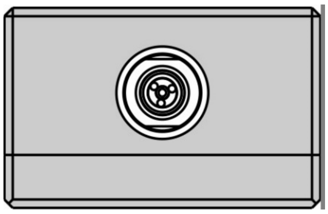
The Modules connectors are shown in figures below.



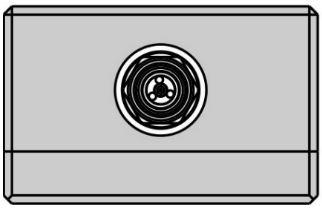
Type N, male



Type N, female

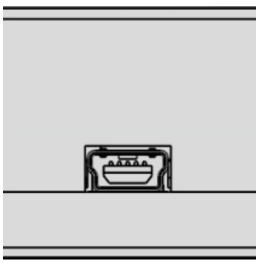


3.5 mm, male



3.5 mm, female

Mini USB Connector (on side panel)



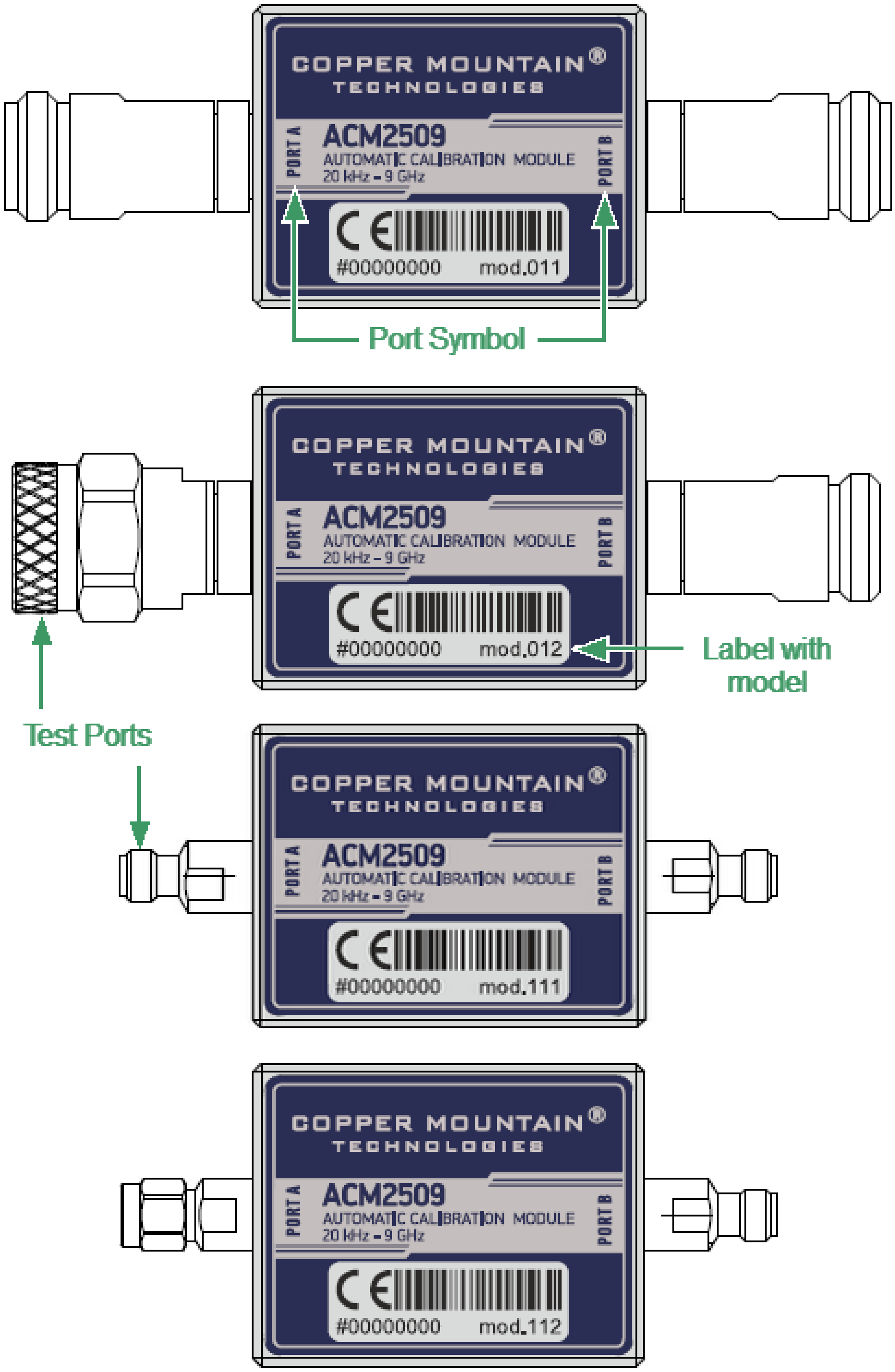
The mini USB connector is located at the side panel of the Module and is intended for the Module connection to the controlling PC. The Module is powered using the USB cable.

Hardware Configurations

Model	Connector type	
	Port A	Port B
ACM2506-011	type N, female	type N, female
ACM2506-012	type N, male	type N, female
ACM2506-111	3.5 mm, female	3.5 mm, female
ACM2506-112	3.5 mm, male	3.5 mm, female

ACM2509

Front panel of different models of ACM2506 are shown in figure below.



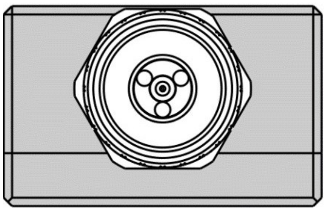
Front panel ACM2509

Parts of Module

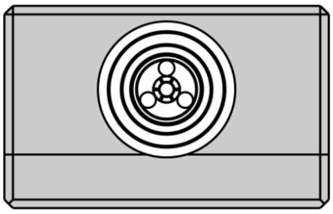
Test Port

The test ports are designed for connection to VNA being calibrated. The VNA connectors, the cross sections of which were calibrated, are referred to as its test ports.

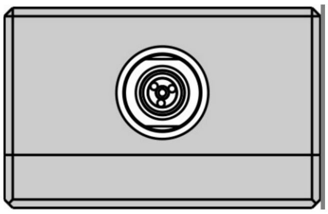
The Modules connectors are shown in figures below.



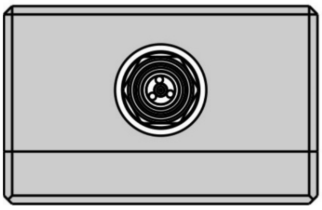
Type N, male



Type N, female

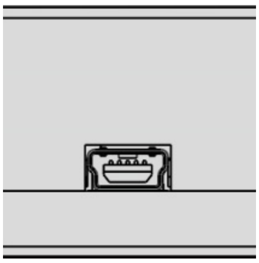


3.5 mm, female



3.5 mm, male

Mini USB Connector (on side panel)



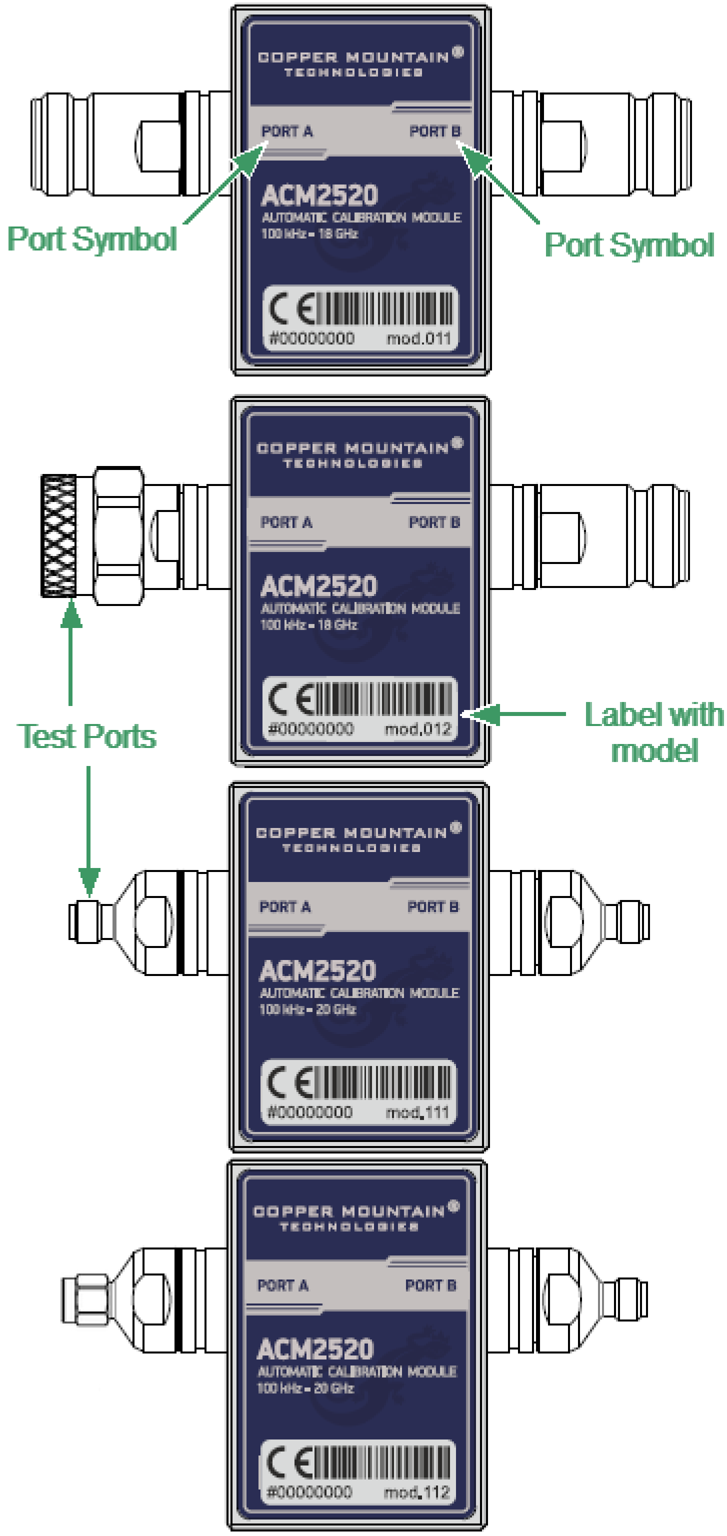
The mini USB connector is located on the side panel of the Module and is intended for the Module connection to the controlling PC. The Module is powered using the USB cable.

Hardware Configurations

Model	Connector type	
	Port A	Port B
ACM2509-011	type N, female	type N, female
ACM2509-012	type N, male	type N, female
ACM2509-111	3.5 mm, female	3.5 mm, female
ACM2509-112	3.5 mm, male	3.5 mm, female

ACM2520

The front panels of the different models of ACM2520 are shown in the figure below.



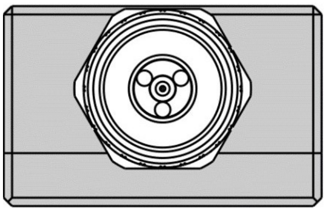
Front panel ACM2520

Parts of Module

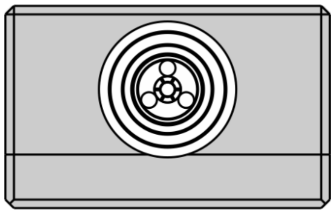
Test Port

The test ports are designed for connection to VNA being calibrated. The VNA connectors, the cross sections of which were calibrated, are referred to as its test ports.

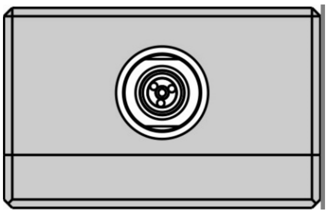
The Modules connectors are shown in figures below.



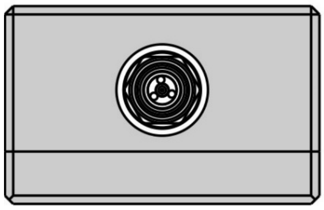
Type N, male



Type N, female

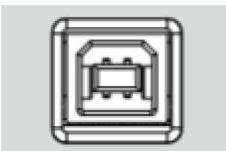


3.5 mm, female



3.5 mm, male

Connector (on side panel)



The connector is located on the top of the Module and is intended for the Module connection to the controlling PC. The Module is powered using the USB cable.

LED Status Indicator (on rear panel)

NOTE	LED Status Indicator is located under the label and is visible only during operation.

The LED indicates the following statuses:

- Blinking green and red LED mean testing LED and indicating external power supply voltage presence.

- Red LED indicator means warm-up mode of the Module. The time required for operating mode setting is automatically counted from the moment of the Module connection using USB. If the Module is disconnected during setting and reconnected again, then the countdown counter starts from the beginning.

  Additional red LED may indicate the Module connection loss with the PC. In this case, check the Module connection with software (the **Autocalibration** softkey should be active), if there is no connection, disconnect the USB cable from the Module and repeat the connection.

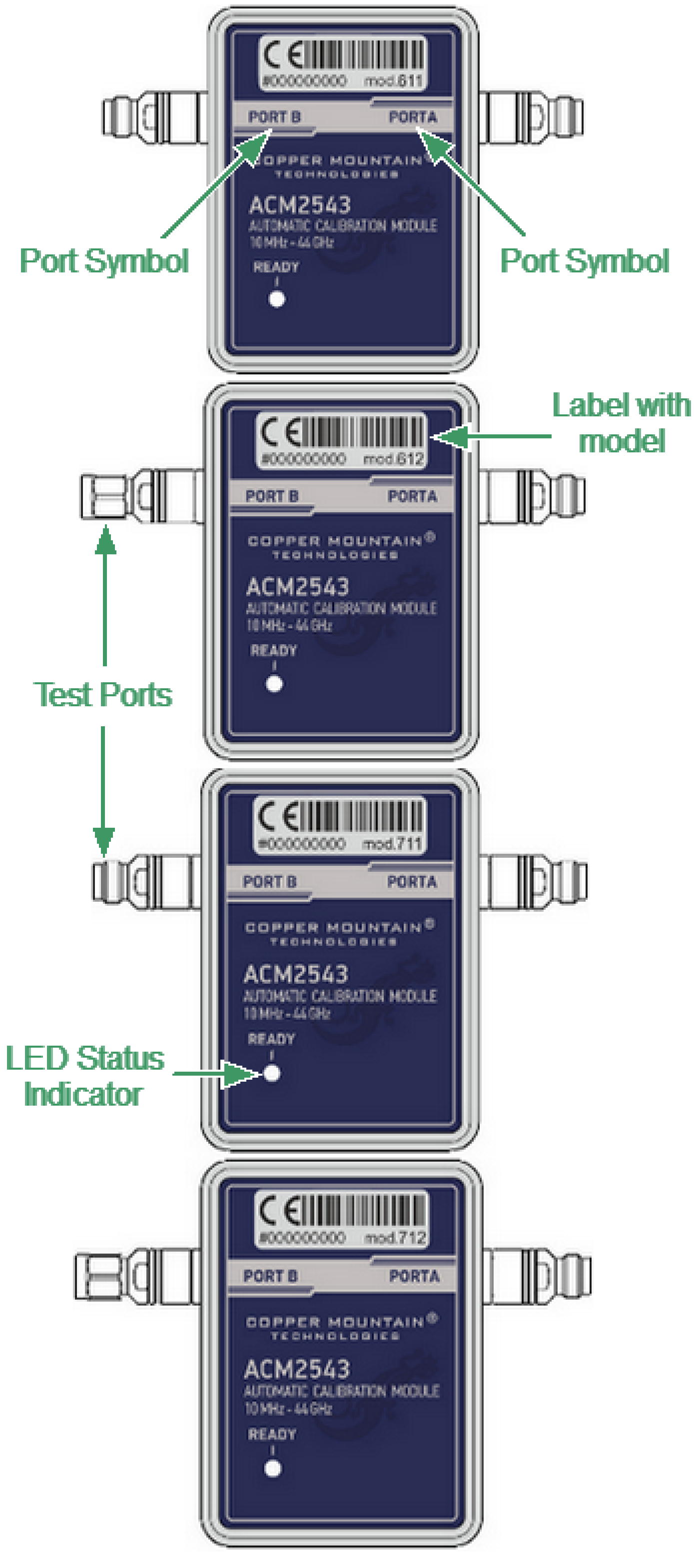
- Green LED indicator means the Module is ready for operation.

Hardware Configurations

Model	Connector type	
	Port A	Port B
ACM2520-011	type N, female	type N, female
ACM2520-012	type N, male	type N, female
ACM2520-111	3.5 mm, female	3.5 mm, female
ACM2520-112	3.5 mm, male	3.5 mm, female

ACM2543

The rear panels of the different models of ACM2543 are shown in the figure below.

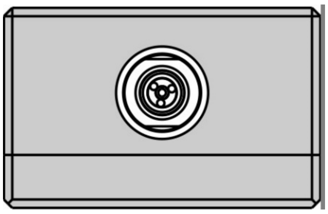


Rear panel ACM2543

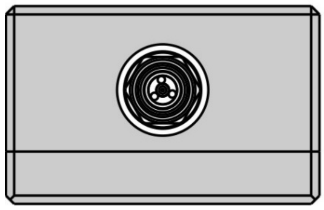
Parts of Module

Test Port

The test ports are designed for connection to VNA being calibrated. The VNA connectors, the cross sections of which were calibrated, are referred to as its test ports. The Modules connectors are shown in figures below.



2.4 mm (2.92 mm), female



2.4 mm (2.92 mm), male

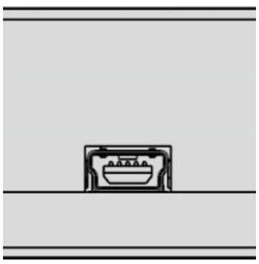
LED Status Indicator

The LED indicates the following statuses:

- Blinking green and red LED mean testing LED and indicating external power supply voltage presence.
- Red LED indicator means warm-up mode of the Module. The time required for operating mode setting is automatically counted from the moment of the Module connection using USB. If the Module is disconnected during setting and reconnected again, then the countdown counter starts from the beginning.

  Additional red LED may indicate the Module connection loss with the PC. In this case, check the Module connection with software (the **Autocalibration** softkey should be active), if there is no connection, disconnect the USB cable from the Module and repeat the connection.
- Green LED indicator means the Module is ready for operation.

Mini USB Connector (on side panel)



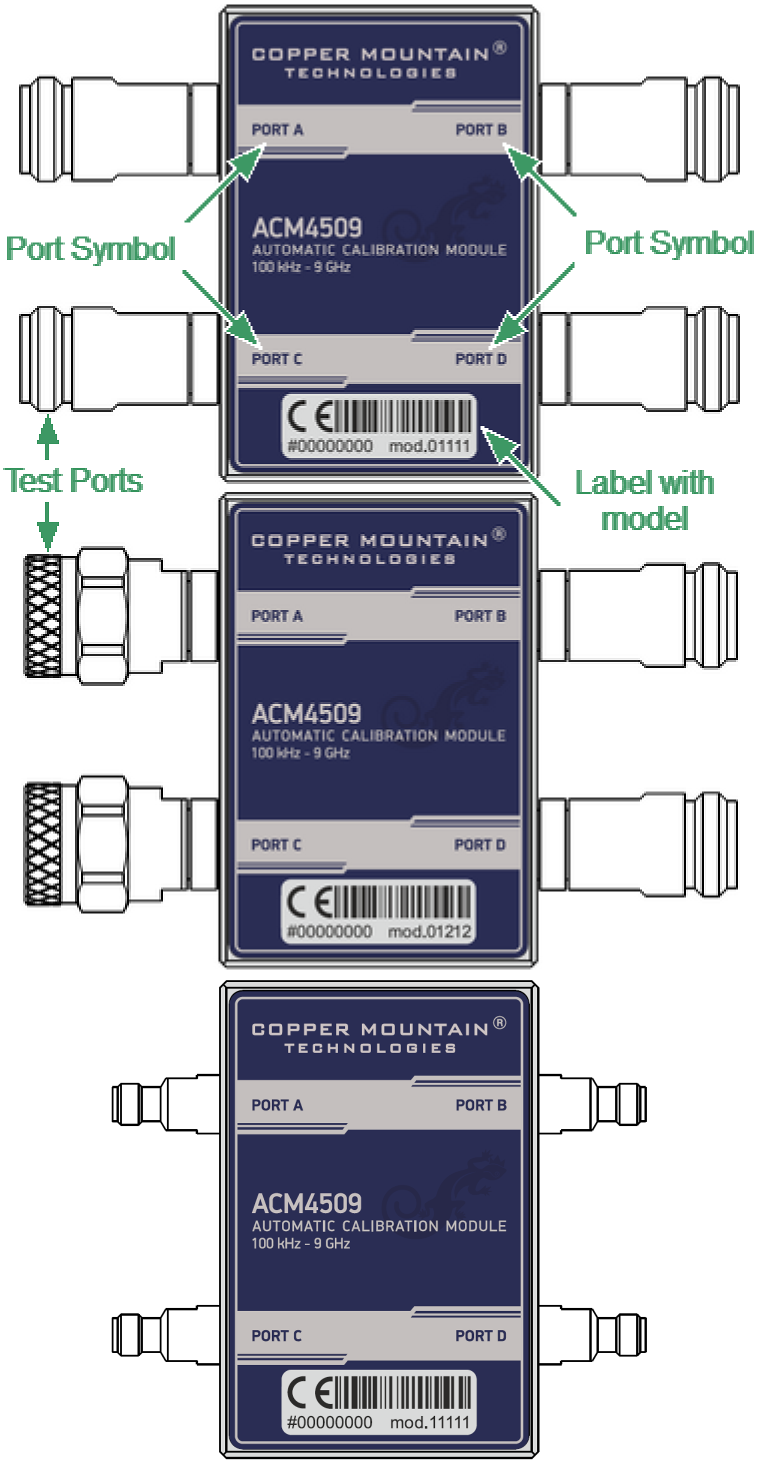
The connector is located on the top of the Module and is intended for the Module connection to the controlling PC. The Module is powered using the USB cable.

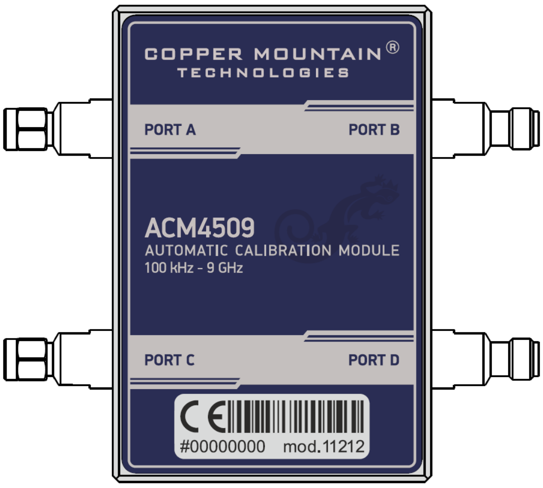
Hardware Configurations

Model	Connector type	
	Port A	Port B
ACM2543-611	2.92 mm, female	2.92 mm, female
ACM2543-612	2.92 mm, male	2.92 mm, female
ACM2543-711	2.4 mm, female	2.4 mm, female
ACM2543-712	2.4 mm, male	2.4 mm, female

ACM4509

The front panels of the different models of ACM4509 are shown in the figure below.





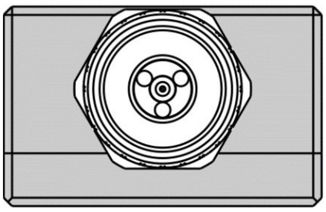
Front panel ACM4509

Parts of Module

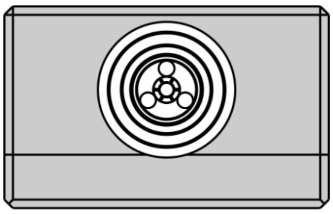
Test Port

The test ports are designed for connection to VNA being calibrated. The VNA connectors, the cross sections of which were calibrated, are referred to as its test ports.

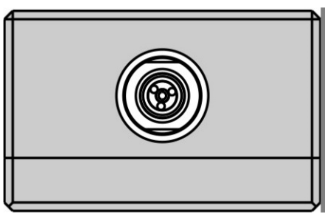
The Modules connectors are shown in figures below.



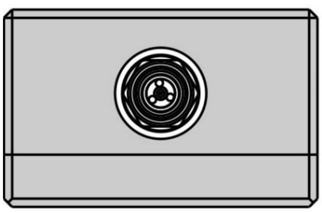
Type N, male



Type N, female

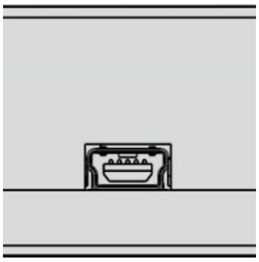


3.5 mm, female



3.5 mm, male

Mini USB Connector (on side panel)



The mini USB connector is located on the bottom of the Module and is intended for the Module connection to the controlling PC. The Module is powered using the USB cable.

LED Status Indicator (on rear panel)

NOTE	LED Status Indicator is located under the label and is visible only during operation.

The LED indicates the following statuses:

- Blinking green and red LED mean testing LED and indicating external power supply voltage presence.
- Red LED indicator means warm-up mode of the Module. The time required for operating mode setting is automatically counted from the moment of the Module connection using USB. If the Module is disconnected during setting and reconnected again, then the countdown counter starts from the beginning.

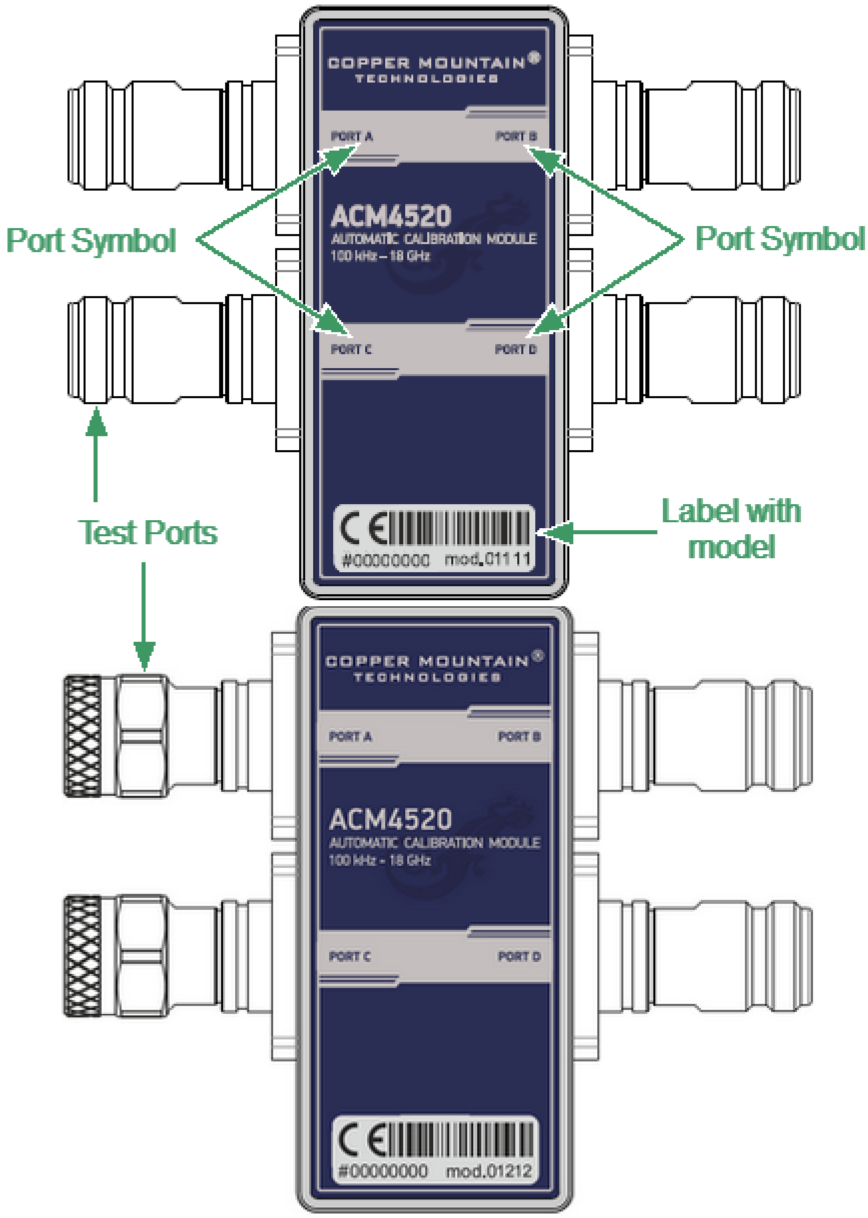
  Additional red LED may indicate the Module connection loss with the PC. In this case, check the Module connection with software (the **Autocalibration** softkey should be active), if there is no connection, disconnect the USB cable from the Module and repeat the connection.
- Green LED indicator means the Module is ready for operation.

Hardware Configurations

Model	Connector type	
	Port A/C	Port B/D
ACM4509-01111	type N, female	type N, female
ACM4509-01212	type N, male	type N, female
ACM509-11111	3.5 mm, female	3.5 mm, female
ACM4509-11212	3.5 mm, male	3.5 mm, female

ACM4520

The front panels of the different models of ACM4520 are shown in the figure below.



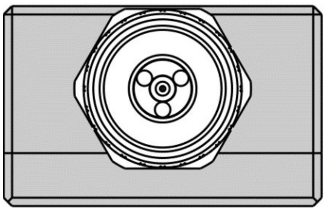
Front panel ACM4520

Parts of Module

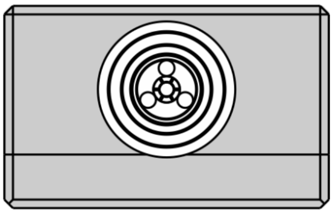
Test Port

The test ports are designed for connection to VNA being calibrated. The VNA connectors, the cross sections of which were calibrated, are referred to as its test ports.

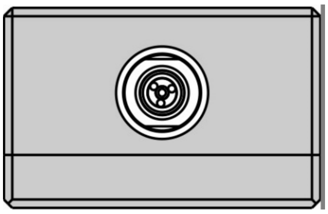
The Modules connectors are shown in figures below.



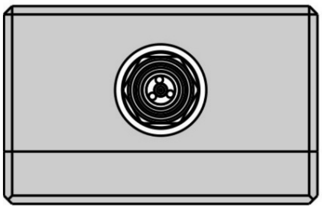
Type N, male



Type N, female

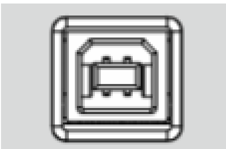


3.5 mm, female



3.5 mm, male

Connector (on side panel)



The connector is located on the bottom of the Module and is intended for the Module connection to the controlling PC. The Module is powered using the USB cable.

LED Status Indicator (on rear panel)

NOTE	LED Status Indicator is located under the label and is visible only during operation.

The LED indicates the following statuses:

- Blinking green and red LED mean testing LED and indicating external power supply voltage presence.
- Red LED indicator means warm-up mode of the Module. The time required for operating mode setting is automatically counted from the moment of the Module connection using USB. If the Module is disconnected during setting and reconnected again, then the countdown counter starts from the beginning.

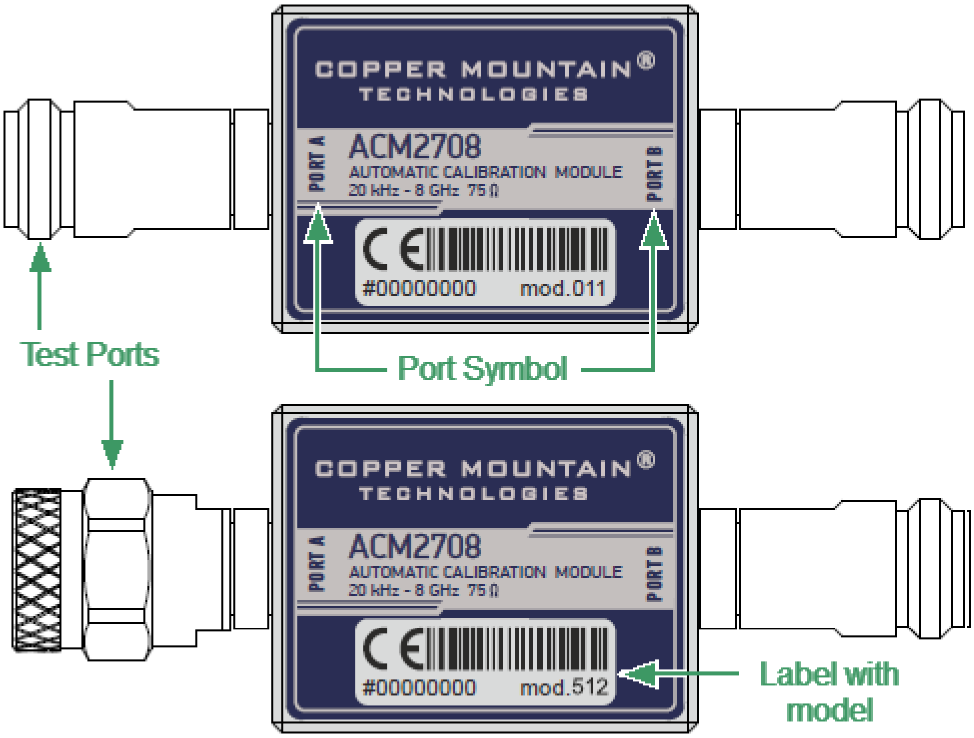
  Additional red LED may indicate the Module connection loss with the PC. In this case, check the Module connection with software (the **Autocalibration** softkey should be active), if there is no connection, disconnect the USB cable from the Module and repeat the connection.
- Green LED indicator means the Module is ready for operation.

Hardware Configurations

Model	Connector type	
	Port A/C	Port B/D
ACM4520-01111	type N, female	type N, female
ACM4520-01212	type N, male	type N, female
ACM4520-11111	3.5 mm, female	3.5 mm, female
ACM4520-11212	3.5 mm, male	3.5 mm, female

ACM2708

The front panels of the different models of ACM2708 are shown in the figure below.



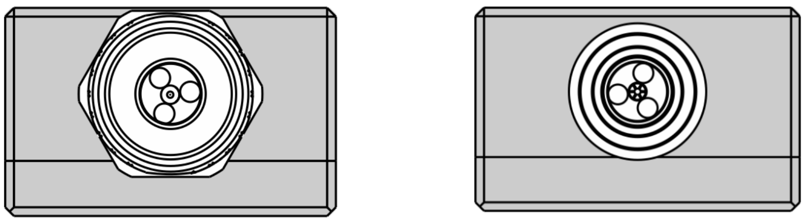
Front panel ACM2708

Parts of Module

Test Port

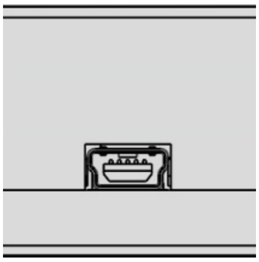
The test ports are designed for connection to VNA being calibrated. The VNA connectors, the cross sections of which were calibrated, are referred to as its test ports.

The Modules connectors are shown in figures below.



Type N 75, male Type N 75, female

Connector (on side panel)



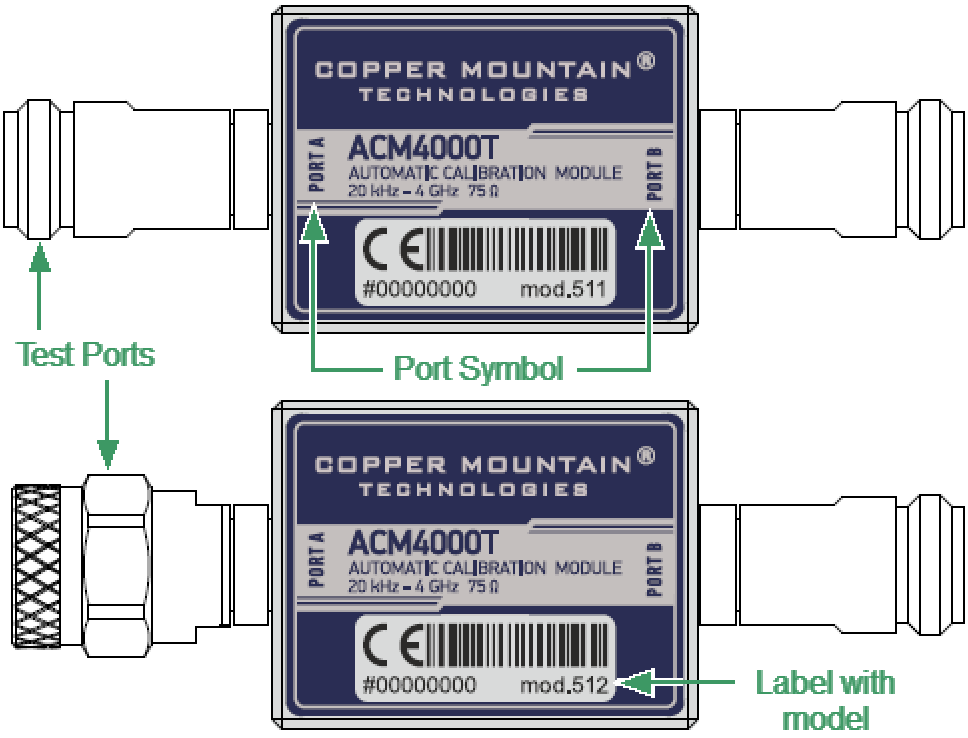
The connector is located on the bottom of the Module and is intended for the Module connection to the controlling PC. The Module is powered using the USB cable.

Hardware Configurations

Model	Connector type	
	Port A	Port B
ACM2708-511	type N 75, female	type N 75, female
ACM2708-512	type N 75, male	type N 75, female

ACM4000T

The front panels of the different models of ACM4000T are shown in the figure below.



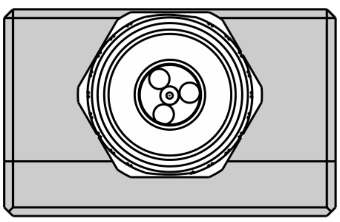
Front panel ACM4000T

Parts of Module

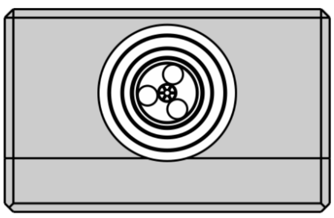
Test Port

The test ports are designed for connection to VNA being calibrated. The VNA connectors, the cross sections of which were calibrated, are referred to as its test ports.

The Modules connectors are shown in figures below.

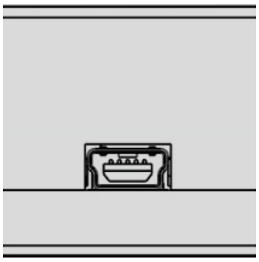


Type N 75, male



Type N 75, female

Connector (on side panel)



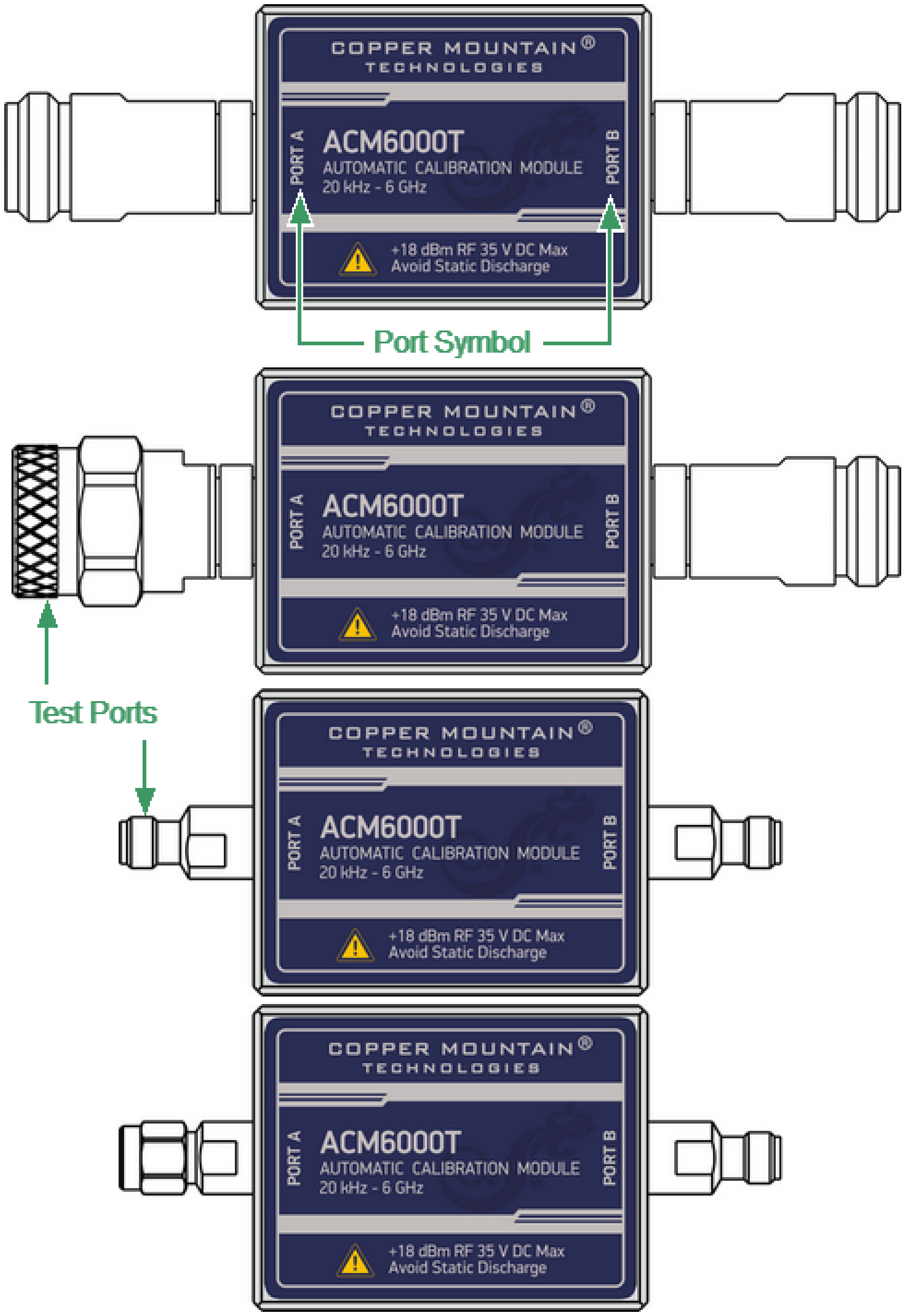
The connector is located on the bottom of the Module and is intended for the Module connection to the controlling PC. The Module is powered using the USB cable.

Hardware Configurations

Model	Connector type	
	Port A	Port B
ACM4000T-511	type N 75, female	type N 75, female
ACM4000T-512	type N 75, male	type N 75, female

ACM6000T

The front panels of the different models of ACM6000T are shown in the figure below.



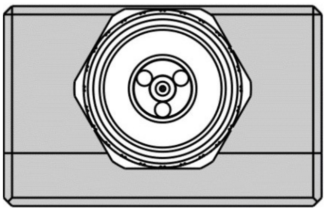
Front panel ACM6000T

Parts of Module

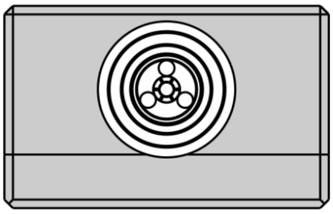
Test Port

The test ports are designed for connection to VNA being calibrated. The VNA connectors, the cross sections of which were calibrated, are referred to as its test ports.

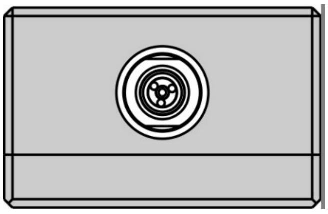
The Modules connectors are shown in figures below.



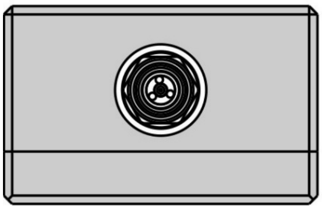
Type N, male



Type N, female

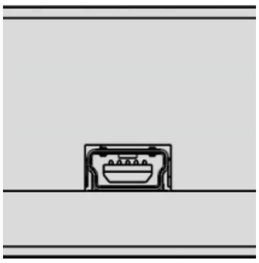


3.5 mm, female



3.5 mm, male

Connector (on side panel)



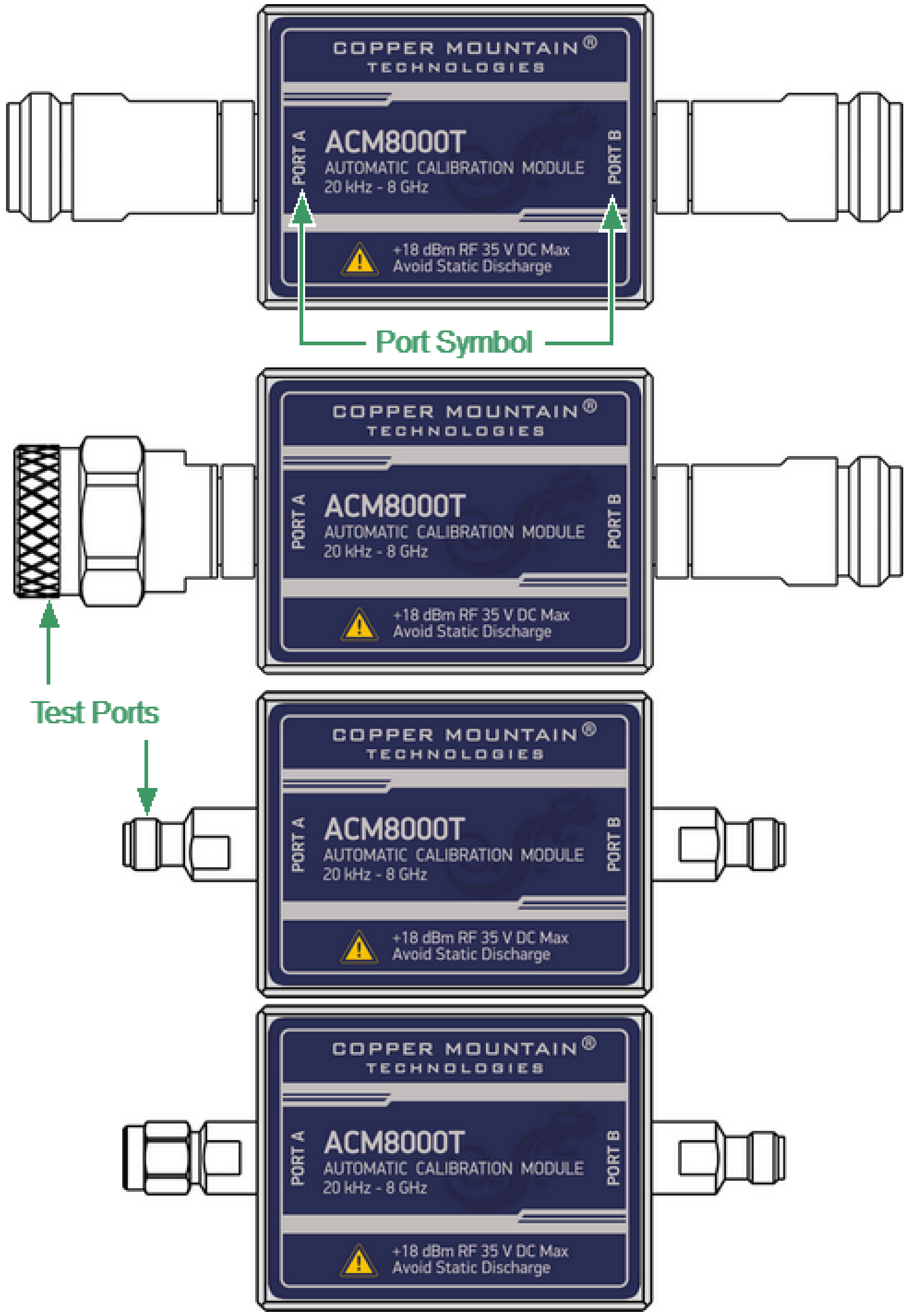
The connector is located on the bottom of the Module and is intended for the Module connection to the controlling PC. The Module is powered using the USB cable.

Hardware Configurations

Model	Connector type	
	Port A	Port B
ACM6000T-011	type N, female	type N, female
ACM6000T-012	type N, male	type N, female
ACM6000T-111	3.5 mm, female	3.5 mm, female
ACM6000T-112	3.5 mm, male	3.5 mm, female

ACM8000T

The front panels of the different models of ACM8000T are shown in the figure below.



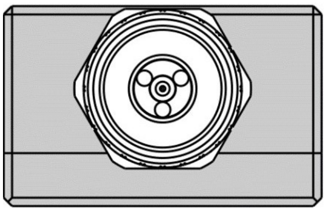
Front panel ACM8000T

Parts of Module

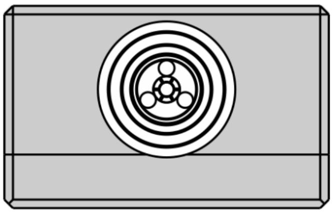
Test Port

The test ports are designed for connection to VNA being calibrated. The VNA connectors, the cross sections of which were calibrated, are referred to as its test ports.

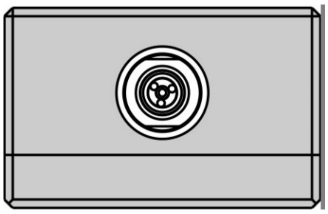
The Modules connectors are shown in figures below.



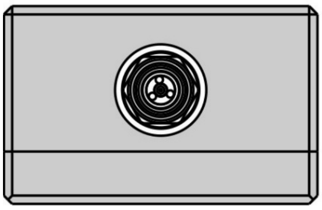
Type N, male



Type N, female

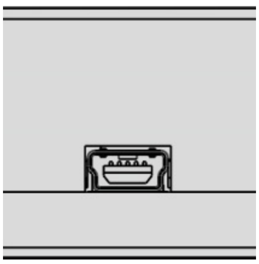


3.5 mm, female



3.5 mm, male

Connector (on side panel)



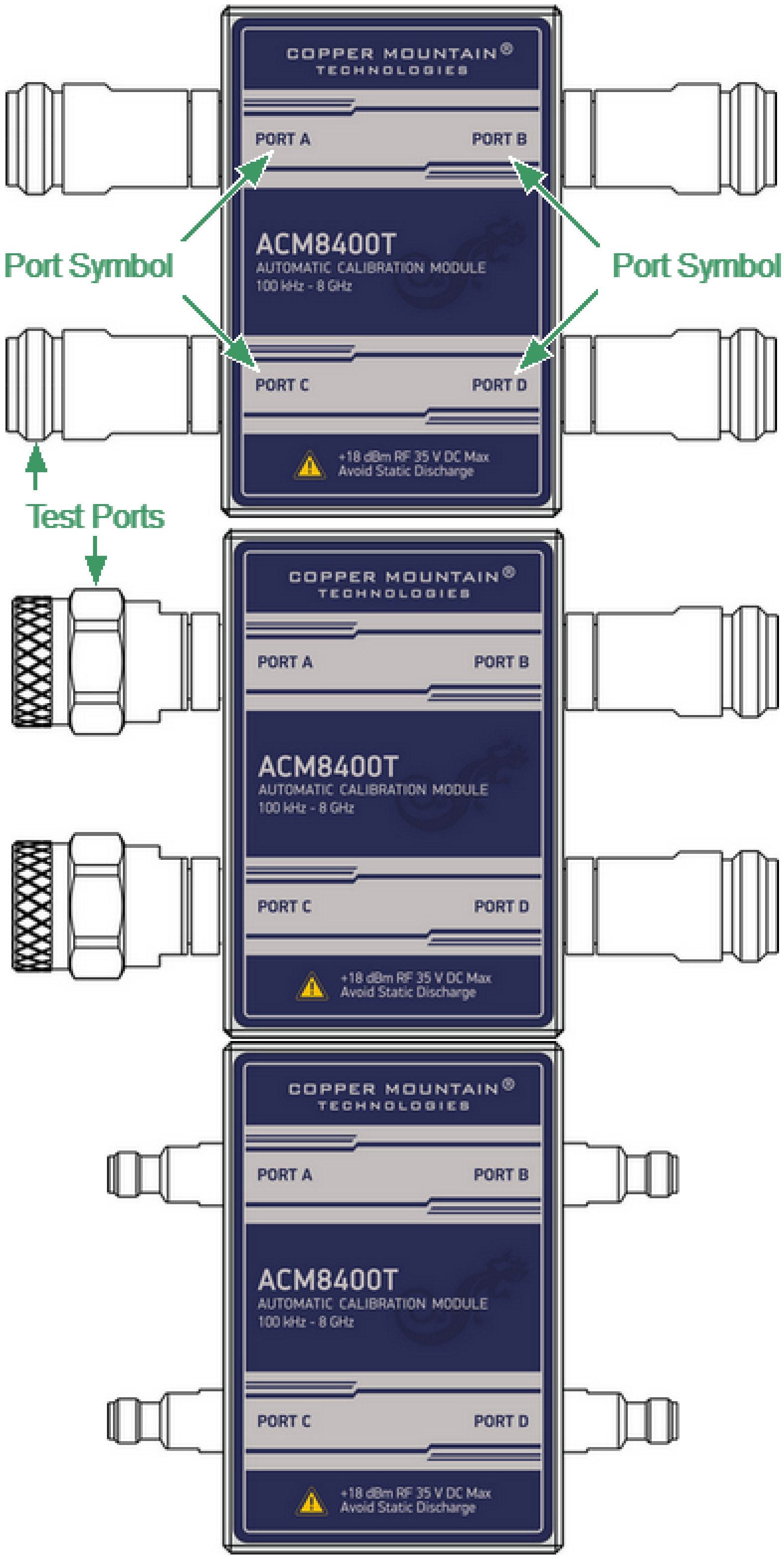
The connector is located on the bottom of the Module and is intended for the Module connection to the controlling PC. The Module is powered using the USB cable.

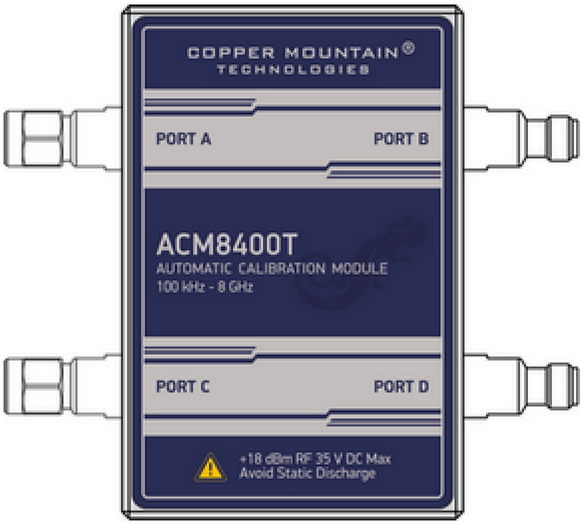
Hardware Configurations

Model	Connector type	
	Port A	Port B
ACM8000T-011	type N, female	type N, female
ACM8000T-012	type N, male	type N, female
ACM8000T-111	3.5 mm, female	3.5 mm, female
ACM8000T-112	3.5 mm, male	3.5 mm, female

ACM8400T

The front panels of the different models of ACM8400T are shown in the figure below.





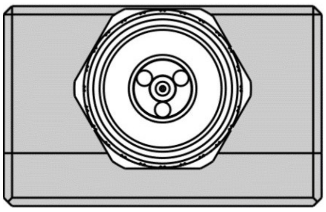
Front panel ACM8400T

Parts of Module

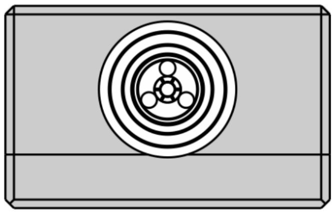
Test Port

The test ports are designed for connection to VNA being calibrated. The VNA connectors, the cross sections of which were calibrated, are referred to as its test ports.

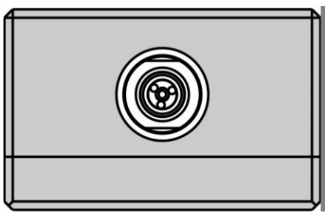
The Modules connectors are shown in figures below.



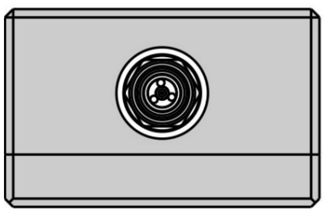
Type N, male



Type N, female

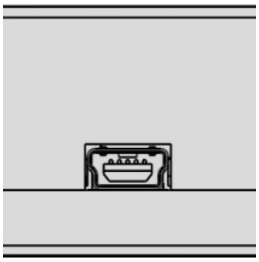


3.5 mm, female



3.5 mm, male

Connector (on side panel)



The connector is located on the bottom of the Module and is intended for the Module connection to the controlling PC. The Module is powered using the USB cable.

LED Status Indicator (on rear panel)

NOTE	LED Status Indicator is located under the label and is visible only during operation.

The LED indicates the following statuses:

- Blinking green and red LED mean testing LED and indicating external power supply voltage presence.
- Red LED indicator means warm-up mode of the Module. The time required for operating mode setting is automatically counted from the moment of the Module connection using USB. If the Module is disconnected during setting and reconnected again, then the countdown counter starts from the beginning.

  Additional red LED may indicate the Module connection loss with the PC. In this case, check the Module connection with software (the **Autocalibration** softkey should be active), if there is no connection, disconnect the USB cable from the Module and repeat the connection.
- Green LED indicator means the Module is ready for operation.

Hardware Configurations

Model	Connector type	
	Port A/C	Port B/D
ACM8400T-01111	type N, female	type N, female
ACM8400T-01212	type N, male	type N, female
ACM8400T-11111	3.5 mm, female	3.5 mm, female
ACM8400T-11212	3.5 mm, male	3.5 mm, female

Protective Housing

The protective housing is designed to protect the test ports and the USB connector of the automatic calibration module (ACM) from mechanical influences.

The protective housing is removable. The collapsible design allows for quick installation.

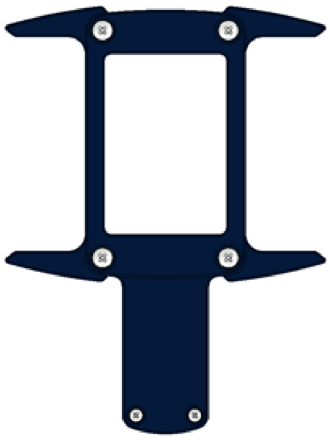
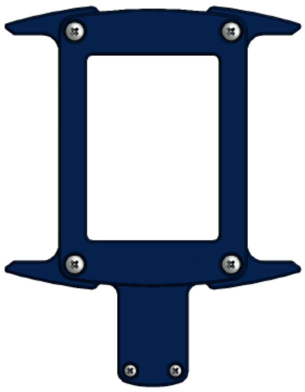
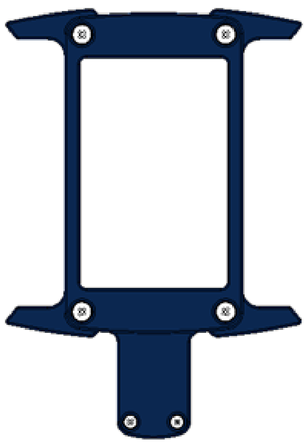
The protective housing is non-repairable.

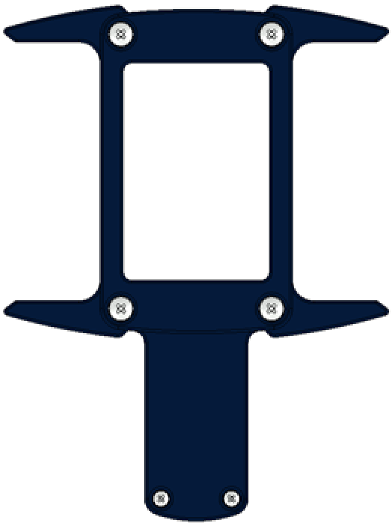
NOTE	The protective housing is not intended for use in extreme environments. Do not bend or stretch the protective housing during use.

The appearance of the protective cover is determined by the modification of the module (See table below).

ACM Protective Housing

Housing Model	Compatible ACM models
ACM2509 	ACM2506-111, ACM2506-112, ACM2509-111, ACM2509-112, ACM6000T-111,ACM6000T-112, ACM8000T-111, ACM8000T-112
ACM2509 	ACM2506-011, ACM2506-012, ACM2509-011, ACM2509-012, ACM2708-011, ACM2708-111, ACM6000T-011,ACM6000T-012, ACM8000T-011, ACM8000T-012, ACM4000T-511, ACM4000T-512

Housing Model	Compatible ACM models
ACM2520 	ACM2520-011, ACM2520-012, ACM2520-111, ACM2520-112
ACM2543 	ACM2543-611, ACM2543-612, ACM2543-711, ACM2543-712
ACM4509 	ACM4509-01111, ACM4509-01212, ACM4509-11111, ACM4509-11212, ACM84000T-01111, ACM84000T-01212, ACM84000T-11111, ACM84000T-11212

Housing Model	Compatible ACM models
ACM4520 	ACM4520-01111, ACM4520-01212, ACM4520-11111, ACM4520-11212

Delivery Kit

The delivery kit for the Module is represented in table below.

Name	Quantity, pcs
Automatic calibration module	1
USB cable	1
Envelope with ACM certificate of calibration and statement of calibration due date	1
Protective housing	1
1. A specific model of Module is selected in the order. 2. The operating manual is not included in the delivery kit , and can be accessed at www.coppermountaintech.com. 3. The protective housing can be ordered separately.	

NOTE

Use the protective housing to protect the test port and USB connector of the Module from mechanical influences (see Protective Housing).

Specifications

The specifications of each Module can be found in its datasheet.

Measurement Capabilities

The VNA software controlling the Module features a wide range of functions. They are briefly described below. See the VNA operating manual for more detailed information.

Automatic Calibration

Calibration	Calibration of a test setup (which includes the VNA, cables, and adapters) significantly increases the accuracy of measurements. Calibration allows for correction of errors caused by imperfections in the measurement system: system directivity, source and load match, tracking, and isolation.
Automatic calibration of VNA	The Module enables calibration in one click. The calibration is performed fully automatically, including switching between different module states, their measurements, and calibration coefficients calculation, as the software uses the data stored in the Module memory.
Calibration methods	All Modules support the following calibration methods: • Full one-port calibration. • One-path two-port calibration.
Full one-port calibration	The method of calibration performed for one-port reflection measurements. It ensures high accuracy.
One-path two-port calibration	The method of calibration performed for reflection and one-way transmission measurements. For example, for measuring S11 and S21 only. It ensures high accuracy for reflection measurements, and reasonable accuracy for transmission measurements.

Unknown thru	The usage of a reciprocal two-port device with loss values of no more than 10 dB for full -port calibration enables correction of VNA parameters for measuring parameters of non-insertion devices. Non-insertion devices are the devices that have same-gender connectors of any type, and different-gender or same-gender connectors of different types. The Module memory stores S-parameters of the thru which are used for calibration coefficients calculation. The said parameters are not applied for the Unknown Thru algorithm.

Characterization

Characterization	Characterization is a table of S-parameters of all the states of the Module switches, stored in its memory. The Module has two memory sections. The first one is write-protected and contains factory characterization. The second memory section allows to store up to three user characterizations. Before calibration, it is possible to select factory characterization or one of the user characterizations.
Factory characterization	Factory characterization is performed during the Module manufacturing. The factory characterization data is stored in the write-protected section of the Module memory.
User characterization	The user characterization option is provided for saving new S-parameters of the Module after connecting adapters to its ports. Up to three different characterizations can be created. The user characterization can be performed using the VNA software. The characterization data is stored in the Module memory section, which can be overwritten.

Automatic Orientation

Orientation	Orientation refers to the Module ports in relation to the test ports of the VNA. While the VNA ports are indicated by numbers, the Module ports are indicated by the letters A, B, C and D.
Orientation method	Manual or automatic orientation method can be selected.
Automatic orientation	For automatic orientation, the VNA software determines the Module orientation each time prior to its calibration or characterization.

Thermal Compensation

Thermal compensation	Thermal compensation is a software function of S-parameters correction based on known temperature dependence data and the temperature sensor data inside the Module. Temperature dependence of each Module with factory characterization is determined during its manufacture and stored in its memory. It is possible to enable or disable thermal compensation function.
Thermal compensation of user characterization	Thermal compensation of user characterization is based on coefficients obtained during the Module manufacture. If the operating frequency range and/or the number of frequency points of the user and factory characterization are not the same, linear interpolation of thermal compensation coefficients is used for user characterization data.

Confidence Check

Confidence check	The confidence check is a test of the current calibration, performed either by the Module, or by any other method. The confidence check features simultaneous indication of attenuator S-parameters measured and stored in the Module memory. Math (division) function for data and memory is used for a detailed comparison.

Automation

Operating modes	The Module is controlled using the USB interface. CMT's VNA software or VISA library must be installed at the controlling PC. The VISA comprehensive library allows controlling measurement equipment in almost all programming languages, i.e. C/C++, Visual Basic, MATLAB, LabVIEW, etc. The Module features the USBTMC USB488 standard control protocol. The Programming Manual includes descriptions of commands used for controlling.

Principle of Operation

The Module contains several different transmission and reflection impedance states, as well as electronic changeover switches, two or four RF connectors, and a USB connector. RF connectors are intended for connecting to VNA test ports, and a USB connector is intended for controlling.

Module	States
ACM2506, ACM2509, ACM2708, ACM4000T, ACM6000T	6 reflection states (three for each port), a THRU, and an attenuator.
ACM2520	8 reflection states (four for each port), a THRU, and an attenuator.
ACM2543, ACM8000T	10 reflection states (five for each port), a THRU, and an attenuator.
ACM4509, ACM8400T	16 reflection states (four for each port), a THRU, and an attenuator.
ACM4520	12 reflection states (three for each port), a THRU, and an attenuator.

Calibration is performed by automatically connecting internal transmission and reflection impedance states to the VNA test ports.

Calibration allows determining systematic errors according to the VNA model. The data obtained after calibration is used to correct S-parameter measurement results to increase measurement accuracy.

Block diagrams of Modules are represented in Module Block Diagrams.

Types of Calibration Standards

Calibration standards are physical devices with known parameters used for VNA calibration, with the purpose of calculating systematic errors and further correcting the measurement results.

OPEN, SHORT, and LOAD are the reflection standards, and THRU is the transmission standard (transmission connection).

The Module includes four types of calibration standards:

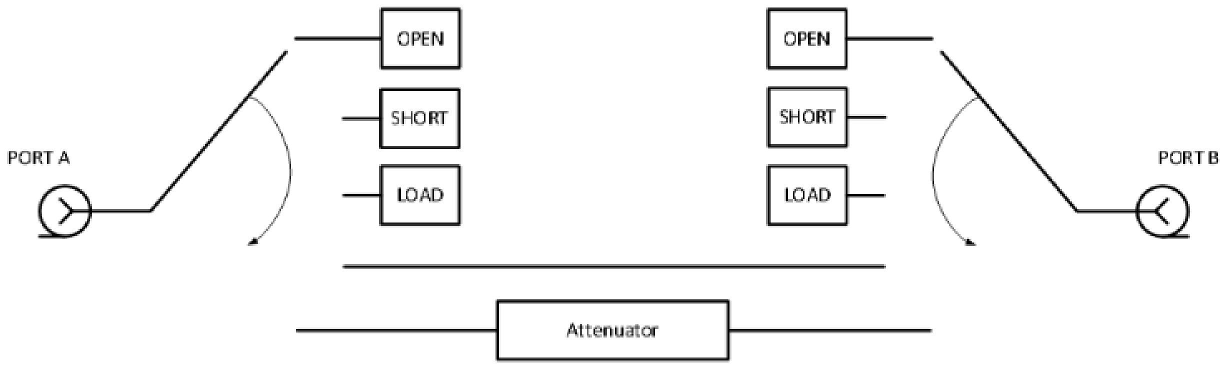
- OPEN
- SHORT
- LOAD
- THRU

Attenuator

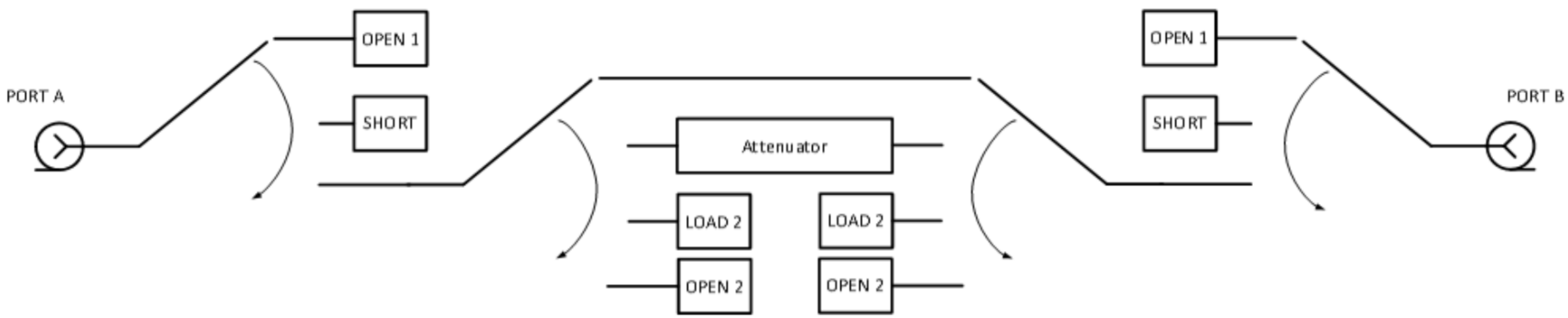
The Module features additional attenuator state, which is not used during calibration. The attenuator is used for checking calibration quality using a special confidence check function, which allows for comparing of the measured S-parameters of attenuator with the parameters stored in the Module memory.

Module Block Diagrams

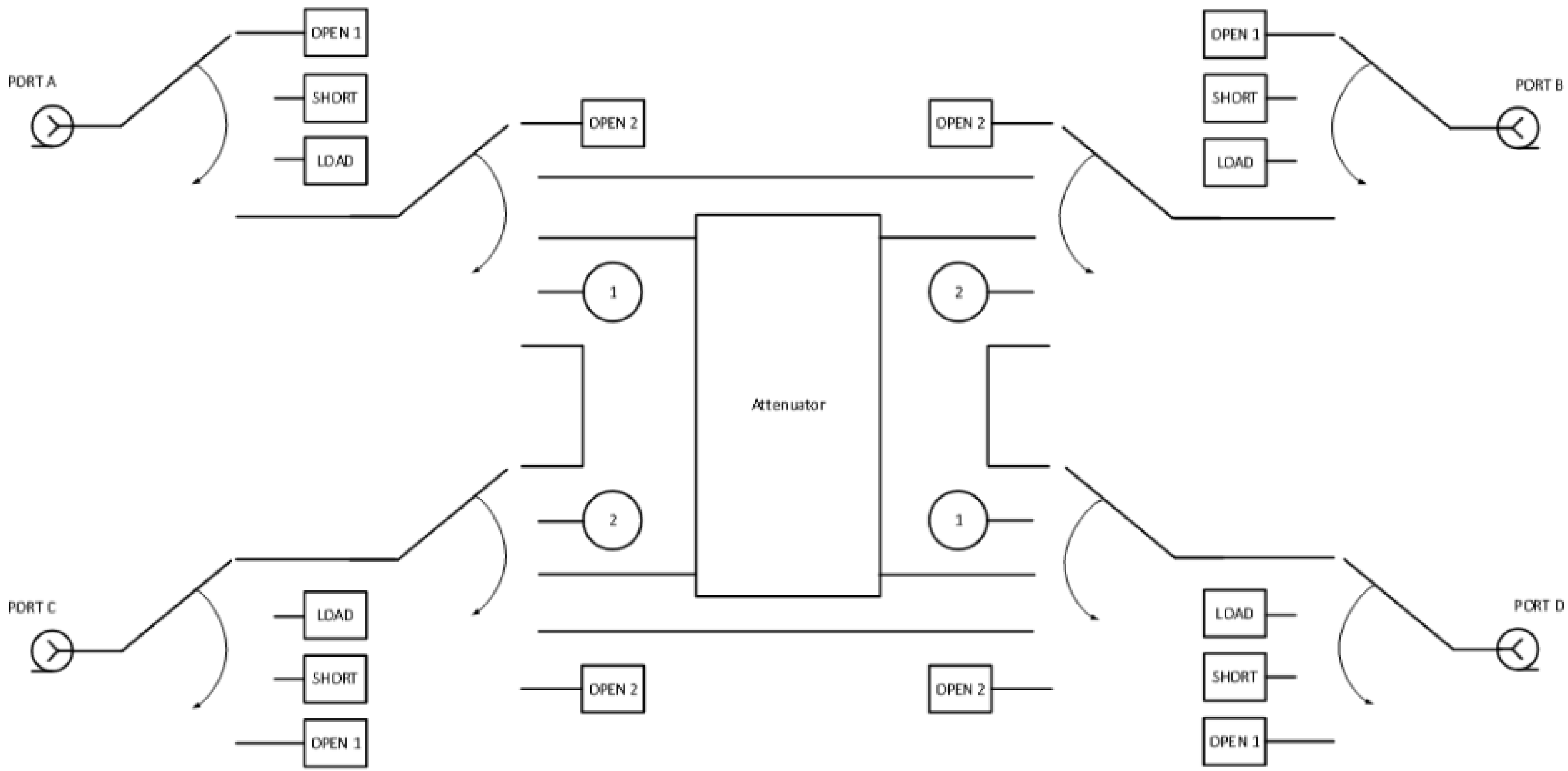
Module block diagrams are shown in figures below.



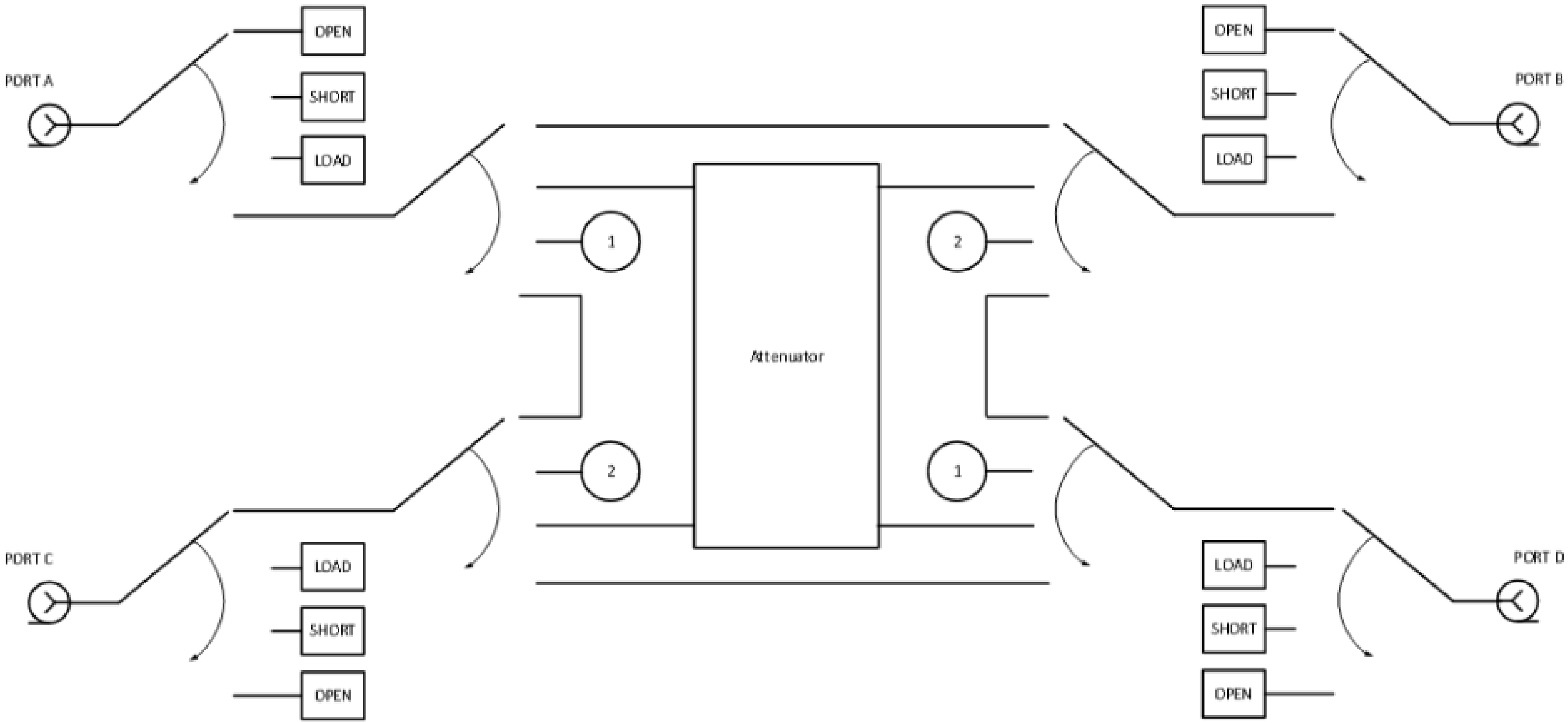
Block diagram of ACM2506 and ACM2509



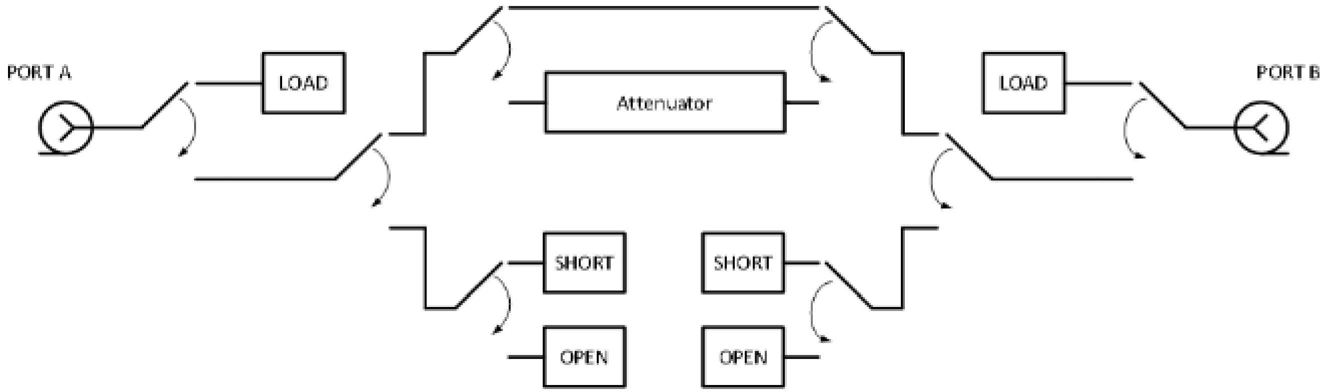
Block diagram of ACM2520



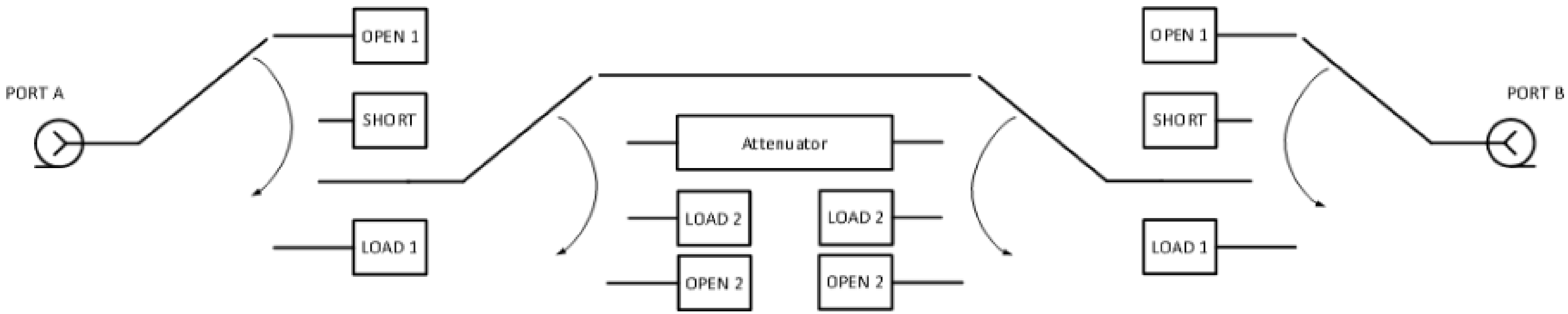
Block diagram of ACM4509 and ACM8400T



Block diagram of ACM4520



Block diagram of ACM2708, ACM4000T and ACM6000T



Block diagram of ACM8000T and ACM2543

Preparation for Use

Unpack the Module and other accessories.

CAUTION	Please keep packaging to safely ship the instrument for annual calibration!

The following section describes the process of preparing the ACM for use:

- Operating Restrictions.
- Installation.
- Software.

Operating Restrictions

The accuracy of calibration using the Module largely depends on proper handling of the Module while preparing it for use. Keep all connectors clean and undamaged to increase the Module's service life. Dirty or damaged connector can deteriorate accuracy characteristics and materially affect the VNA calibration results.

Before starting operation, perform the following activities to prevent the Module damage:

- Visually inspect the connectors, the Module housing, and the USB cable from the delivery kit for damages and contamination. If foreign particles are detected on the connectors, perform cleaning according to the procedure in Cleaning Connectors. Do not operate the Module if mechanical connector damage is detected. Damaged Modules should be discarded to prevent further damage of other good connectors.
- Visually inspect the connectors, which will be connected to the Module, for damages and contamination. If foreign particles are detected on the connectors, perform cleaning according to the procedure in Cleaning Connectors.
- If necessary, gauge the connectors using the procedure described in Gauging Connectors, which describes connection of the Module and devices connected to it.

Pay special attention to the connection sequence. Proper connection sequence prevents central and external conductors damage, ensures maximum measurement results repeatability, and excludes the most common VNA measurement error, i.e. bad connection. The recommended connection sequence is shown in Connecting and Disconnecting Devices.

The main cause of measurement accuracy deterioration is the change of ambient conditions between the calibration and DUT measurement. The ambient conditions are described in Ambient Conditions Control.

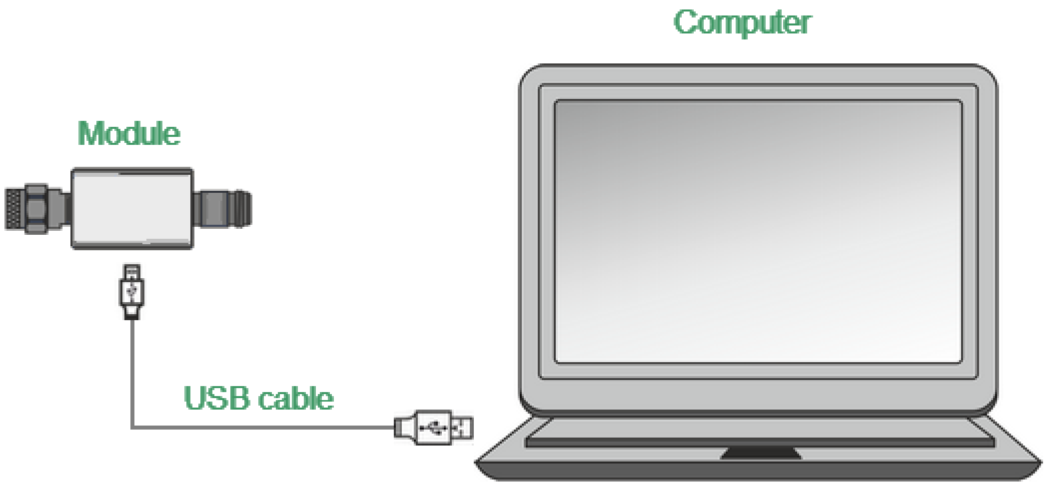
Installation

Unpack the Module and place the Module in the work area.

Take necessary precautions to protect against electrostatic discharge in the work area.

Keep the Module in operating conditions for no less than two hours if it was stored in any other ambient conditions.

Connect the Module using the USB cable. Warm the Module up for no less than 15 minutes. The warm-up connection procedure is shown in the figure below.



Module Connection to PC

Typical Module connection diagrams for VNA calibration are shown in Connection Diagrams.

Software

The Module is controlled by the Copper Mountain Technologies VNA software. Minimum technical requirements to the PC and the description of software installation are described in the VNA Operating Manual.

The VNA software automatically detects the connected Module and makes the Autocalibration menu available. Special Module selection is not generally required.

If the menu is not active:

1. Shut down all the open VNA software windows.
2. Disconnect the Module from the USB cable for one minute, then reinsert the cable.
3. Restart the VNA software, making sure that the VNA software functions properly according to the VNA Operating Manual.
4. Connect the Module again, making sure that the model and serial number match the Module connected.

Driver Installation

The USB driver is automatically installed when the Module is first connected to the USB port.

Operation Procedure

This section describes how to work with the Module:

- Connection diagrams to perform calibration.
- Module work session.
- Parameters setting.

Connection Diagrams

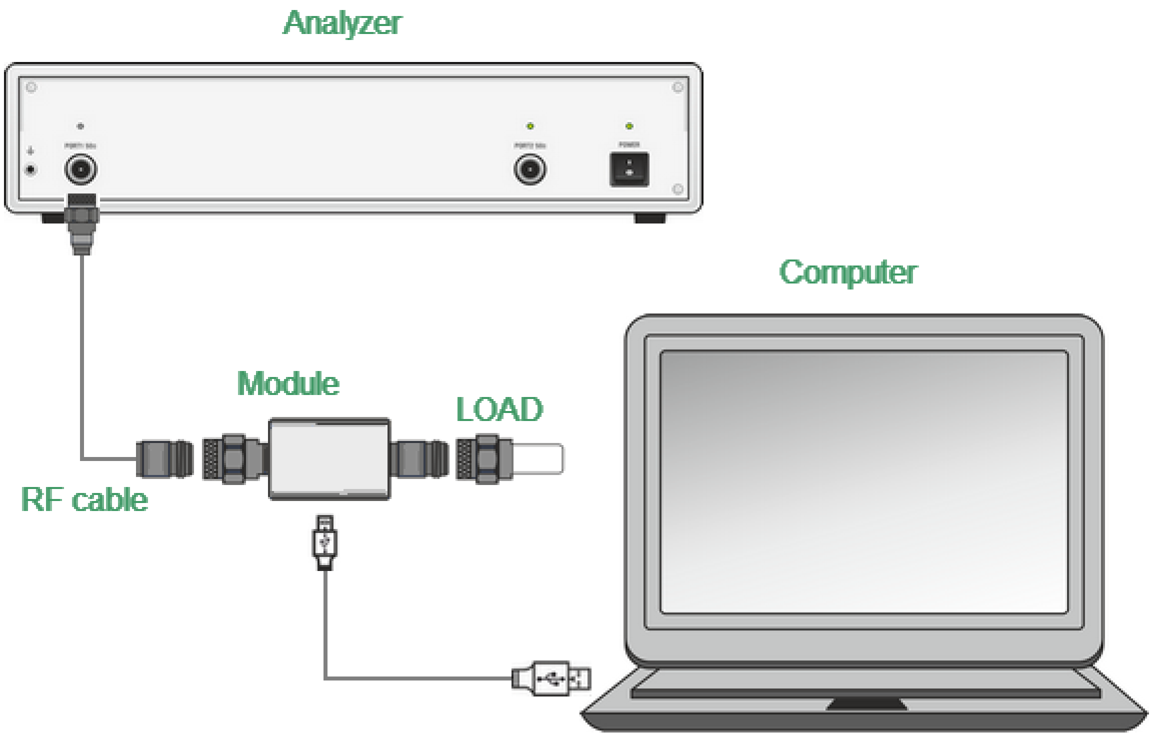
The following are connection diagrams for calibrations:

- Full One-Port CalibrationOne-Path Two-Port

Full One-Port Calibration

In order to perform calibration, it is recommended to connect a LOAD to a free port of the Module. The LOAD is not included in the delivery kit.

Typical connection diagram for full one-port calibration is shown in figure below.



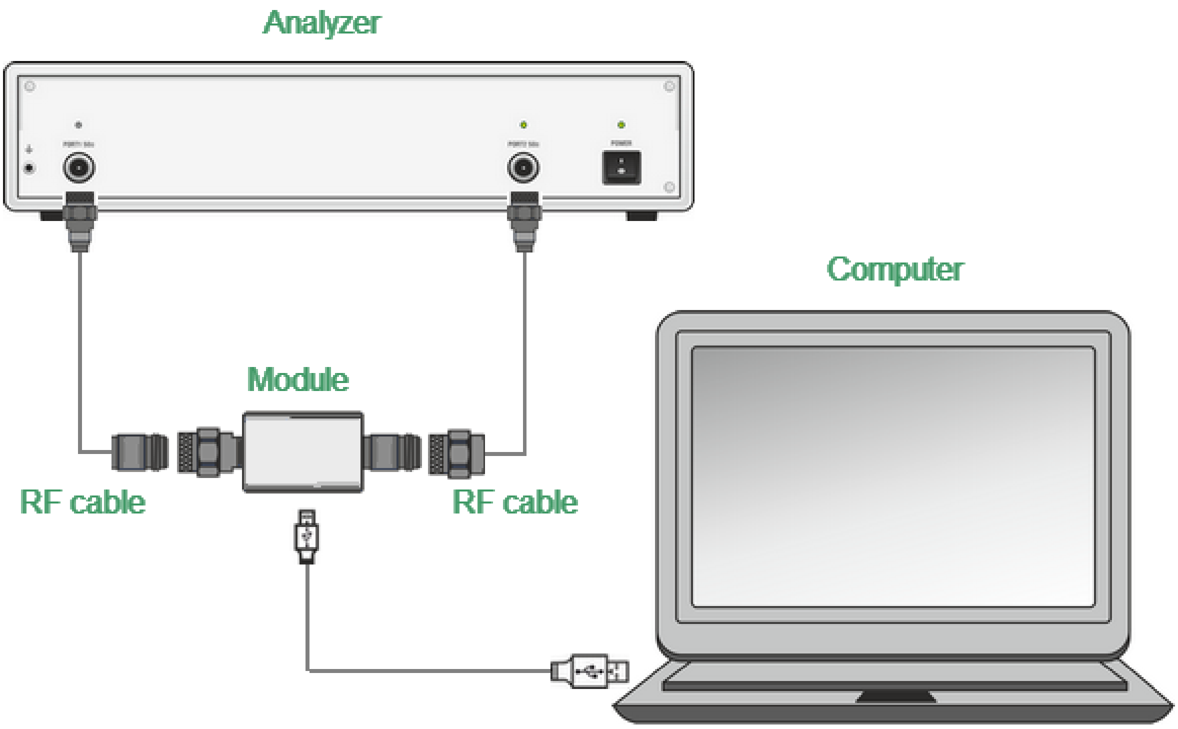
Module Connection Diagram for Performing Full One-port Calibration

To prevent the cable from damage and improve the stability, it is recommended to use additional protection metrology-grade adapters (these adapters are not shown in figure).

WARNING	Use a torque wrench to tighten the male connector nut. Use a spanner to prevent the connected devices from rotation.

One-Path Two-Port

Typical connection diagram for one-path two-port is shown in figure below.



Module Connection Diagram for Performing One-path Two-port and Full Two-port Calibration

To prevent the cable from damage and improve the stability, it is recommended to use additional protection metrology-grade adapters (these adapters are not shown in figure).

WARNING	Use a torque wrench to tighten the male connector nut. Use a spanner to prevent the connected devices from rotation.

Module Work Session

This section includes the example of the Module work session. Perform the following activities to calibrate all types of VNAs:

- Locate the Module at the work site and warm it up for at least 15 minutes.
- Set up the VNA parameters, at which calibration and DUT parameters measurement will be performed.
- Assemble a test setup.
- Connect the Module (typical connection diagrams are shown in Connection Diagrams).
- Perform the required calibration.
- Disconnect the Module and connect the DUT in its place.

Module Preparation for Calibration

Locate the Module on the work bench, switch it on, and warm it up for at least the period of time indicated in the datasheet. If the model used is equipped with an LED status indicator, wait until the LED is green.

WARNING	The technical specifications will correspond to the stated specifications only after the operating mode setup time is over.

Module readiness indication	The VNA software can automatically detect the connected Module. After the Module connection, the VNA software makes the Autocalibration menu available.

Parameters Setting

Before starting measurements and calibration, set up the following VNA parameters:

- Set up default parameters.
- Select the traces and assign measured S-parameters to them.
- Set up the frequency range and the number of frequency points.
- Set up the output power level at no more than -5 dBm.
- Set up the IF bandwidth.

These parameters are set up in the VNA software. The setting procedure is described in detail in the VNA Operating Manual.

Calibration

The following section describes the process of calibrating ACMs.

Module Advantages

Calibration involving the Module has several advantages compared to conventional calibration with a kit of mechanical calibration standards:

- Only one connection required.
- Reduced calibration time.
- Less probability of operator’s mistakes.
- Less wear of VNA test ports connectors.

Measurement Errors

Different measurement errors affect the results of VNA S-parameter measurements. The measurement errors can be divided into two categories:

- Systematic errors.
- Random errors.

Random errors are:

- Noise fluctuations and thermal drift in electronic components.
- Changes in the mechanical dimensions of cables and connectors subject to temperature drift.
- Repeatability of connections and cable bends.

Random errors are unpredictable and hence cannot be estimated and eliminated in calibration. Certain measures can be taken to reduce the random error:

- Proper source power selection.
- Narrower IF bandwidth.
- Constant ambient temperature.
- Proper warm-up time.
- Careful handling of connectors.
- Fewer cable bends after calibration.
- Sage of torque wrench to tighten the male connector nut and spanner to prevent the connected devices from rotation.

Systematic errors occur when the test setup components are not in ideal conditions. They are repeatable, and their characteristics do not change in time. Systematic errors can be calculated, and their value can be reduced mathematically by measurement results correction.

Calibration Types

The Modules enable three types of calibration:

- Full one-port calibration
- One-path two-port calibration

The calibration procedure is described in Calibration Procedure.

Full One-Port Calibration

The three calibration standards are measured in the process of this calibration:

- SHORT
- OPEN
- LOAD

Full one-port calibration features high accuracy.

One-Path Two-Port Calibration

One-path two-port calibration combines full one-port calibration and extended transmission normalization. This calibration type features higher accuracy of measuring frequency response flatness compared to transmission normalization.

One-path two-port calibration requires connection of three calibration standards to the source port, just as in one-port calibration, as well as a connection of the THRU calibration standard between the calibrated source port and the receiver port.

Unknown Thru

UNKNOWN THRU is used in one-path two-port calibration. The calibration type with an UNKNOWN THRU is called SOLR, which refers to Short, Open, Load, Reciprocal.

Any arbitrary reciprocal two-port device with unknown parameters can be used as an UNKNOWN THRU.

There are two basic requirements to the UNKNOWN THRU:

- The first requirement applies to the transmission coefficient of the THRU. It should satisfy the reciprocity condition (S21 = S12), which holds for almost any passive network. Do not use a THRU with a loss higher than 20 dB, as it can reduce the calibration accuracy.
- The second requirement is knowledge of the approximate electrical length of the UNKNOWN THRU within an accuracy of 1/4 of the wavelength at the maximum calibration frequency. This requirement, however, can be omitted if the following frequency step size condition is met:

$$\Delta F < \frac{1}{4 \cdot \tau_0},$$

where τ_0 is a delay of reciprocal two-port device.

In this case, the VNA software can automatically determine electrical length (delay) of a reciprocal two-port device.

A thru, implemented inside the Module using an electronic switch, features loss. Make sure the exact thru parameters are known, or use an UNKNOWN THRU algorithm to obtain the required calibration accuracy.

The Module allows the use of both variants. Its memory stores S-parameters of the thru, which are used for calculation of calibration coefficients. The above parameters are not used if the UNKNOWN THRU algorithm is applied.

Thermal Compensation

Thermal compensation is a software function of the Module parameters correction using the data of internal temperature sensor and data on temperature dependence.

The Module temperature dependence data are the thermal compensation coefficients of magnitude and phase of reflection or transmission coefficients for different Module states stored in its memory.

The compensated magnitude value M_c, dB, is calculated using the following formula:

$$M_c = M \cdot k_m \cdot (T_{char} - T)$$

where M — magnitude before compensation, dB,

k_m — thermal compensation coefficient magnitude, dB/ºC,

T_{char} — temperature at Module characterization, ºC,

T — current temperature inside the Module housing, ºC.

Compensated phase value, P_cº , is calculated using the following formula:

$$P_c = P \cdot k_p \cdot (T_{char} - T) \quad ,$$

where P — phase value before compensation, º

k_p — thermal compensation coefficient phase, º/ºC,

T_{char} — temperature at Module characterization, ºC,

T — current temperature inside the Module housing, ºC,

Temperature dependence of S-parameters of each Module is measured at the factory and stored in its memory.

Thermal compensation can be applied to the factory or user characterization data.

The thermal compensation function can be enabled or disabled.

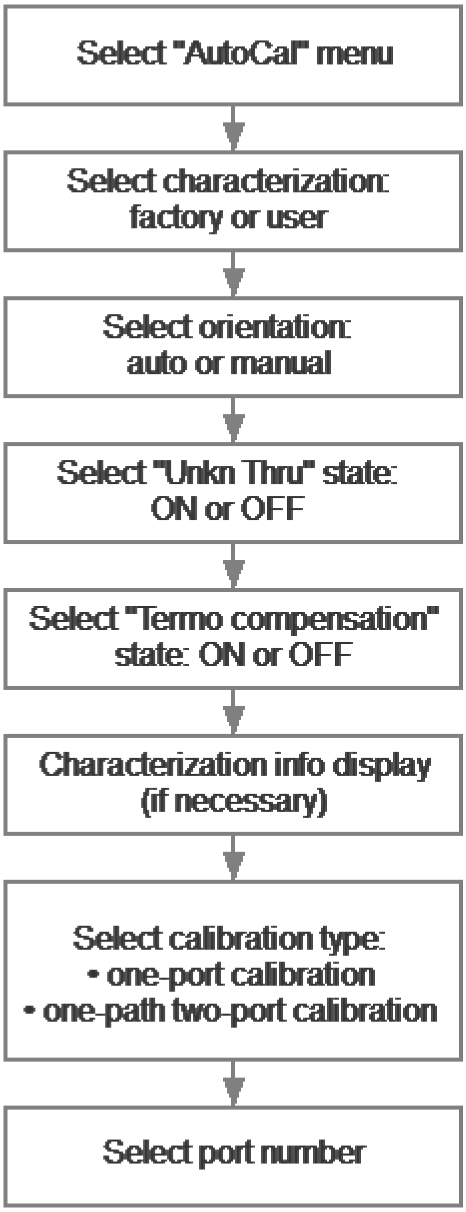
Calibration Procedure

Calibration is performed in fully automatic mode.

The calibration procedure is the following:

1. Press the calibration softkey in the software main menu.
2. Select automatic calibration in the resulting menu. The autocalibration softkey becomes active after the Module connection (typical connection diagrams are shown in Connection Diagrams).
3. Press the characterization softkey.
4. Select factory characterization or one of three user characterizations (user characterization procedure is described in User Characterization Procedure) in the characterization menu.
5. Select the Module orientation method by pressing the orientation softkey.
6. Select the unknown thru algorithm state. The unknown thru algorithm can be either enabled or disabled.
7. Select the thermal compensation function state. The thermal compensation function can be either enabled or disabled.
8. If necessary, display the detailed information on characterization. The information can be displayed by pressing the respective softkey in the autocalibration menu.
9. Select the calibration type: one-port or two-port.
10. Specify the port for full one-port calibration.
11. Wait until calibration is completed.

The automatic calibration algorithm is shown in the figure below.



Autocalibration Algorithm

The calibration will be performed automatically: the standards from the Module set will be connected to VNA in sequence under the VNA software control. Then the calibration coefficients table will be calculated and stored in the VNA memory.

When calibration is completed, certain icons will be indicated in the status bars of reflection and transmission coefficients traces:

- **[F1]** — full one-port calibration.
- **[OP]** — one-path two-port calibration.

Detailed information on calibration using the Module and the names of all softkeys for all VNAs can be found in the VNA Operating and Programming Manual.

Confidence Check

Confidence check is a test of current calibration performed either using the Module or any other method.

The Module features an additional attenuator state that is not used during calibration. The attenuator is intended for checking calibration by means of a special software function, which enables comparison of measured attenuator S-parameters and the values stored in the Module memory.

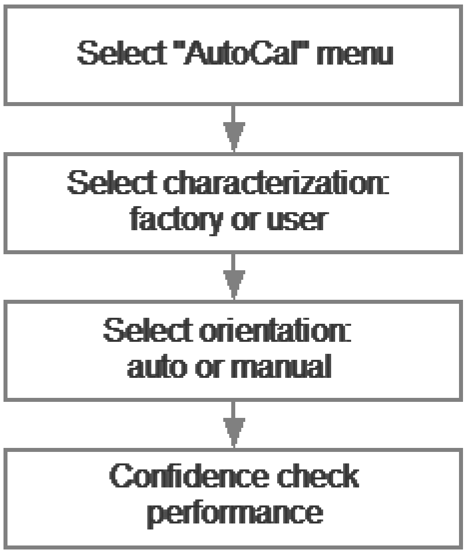
Confidence Check Procedure

1. Press the calibration softkey in the software main menu.
2. Select automatic calibration in the resulting menu.
3. Press the characterization selection softkey in the autocalibration menu.
4. Select factory characterization or one of three user characterizations in the characterization menu.
5. Select the Module orientation method by pressing the orientation softkey in the autocalibration menu. It is recommended to use automatic orientation.
6. Press the «Confidence Check» softkey in the autocalibration menu.
7. Wait until the confidence check is completed.

The confidence check will be performed automatically. Two traces for each S-parameter will be displayed after measurement. The measured parameters will be indicated on the data trace, and the parameters from the Module memory will be indicated on the memory trace.

Compare the data and memory traces to evaluate whether the calibration was successful. Also, the function of math operations with memory traces for a finer trace comparison can be used.

Confidence check algorithm is shown in the figure below.



Algorithm of Confidence Check Using the Module

Detailed information on the Module confidence check and the names of all softkeys for all VNAs can be found in the VNA Operating and Programming Manual.

Automation

The Module supports remote control using third party software. The control function is implemented by means of USB protocol. The VISA library must be installed on the PC for interaction.

The library allows for controlling of measuring equipment in almost any programming language, i.e. C/C++, Visual Basic, MATLAB, LabVIEW, etc. The VISA laboratory supports multiple interfaces and protocols, including USBTMC-USB488 based protocol implemented in the Module.

For detailed information on control functions, see the VNA Operating and Programming Manual.

Maintenance

This section establishes the procedure and rules of maintenance, enabling constant Module operational readiness.

The purpose of Module maintenance is to control its performance parameters and secure its service life.

Maintenance Procedure

The Maintenance Procedure is as follows:

- Maintenance Activities
- Cleaning Connectors
- Gauging Connectors
- Connecting and Disconnecting Devices
- Cleaning and Care of the Protective Housing
- Ambient Conditions Control
- Verification

Maintenance Activities

The Module maintenance includes the following activities:

- Inspection.
- Functional test.

The inspection should be done every time before and after the Module is used.

The inspection comprises:

- Checking components against the delivery kit list.
- Cleaning dust and dirt from external surfaces of the Module. To clean the Module's external surfaces, use dry or slightly wet cloth. Do not clean the Module inside.
- Cleaning connectors as described in Cleaning Connectors.

Functional test should be carries out once per 100 connections.

The functional test includes:

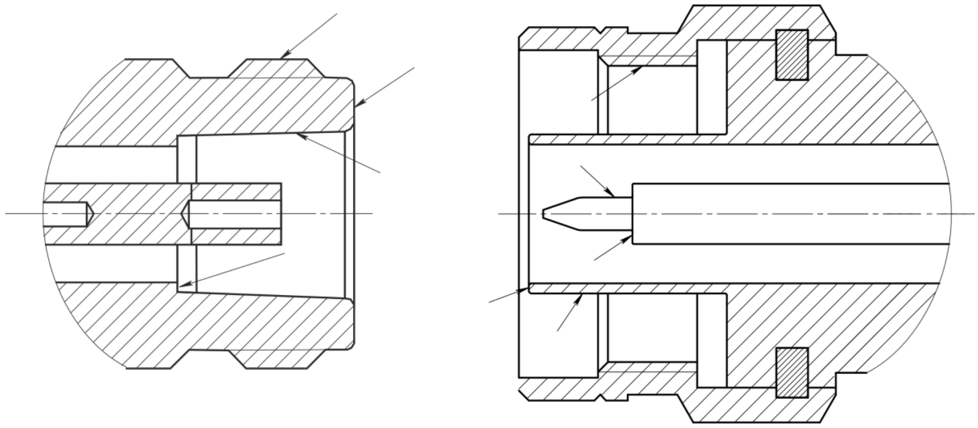
- Inspection.
- Module connectors gauging as described in Gauging Connectors.
- Confidence check.

Cleaning Connectors

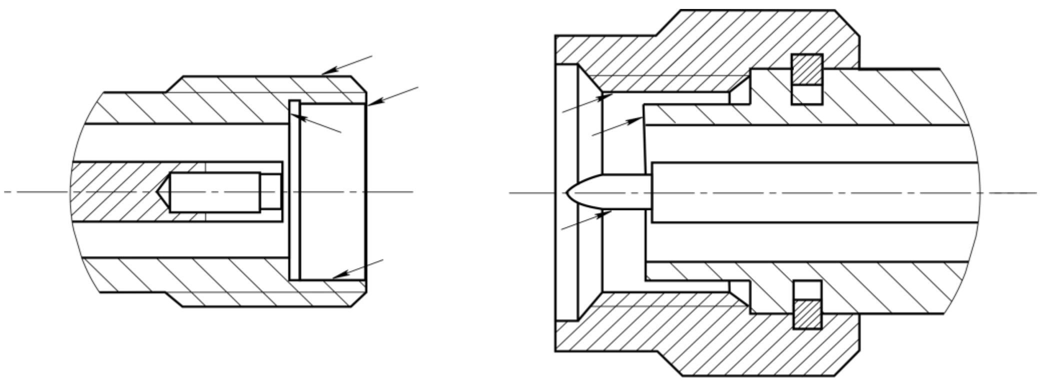
Clean the connectors before and after connecting the Module.

The procedure of cleaning connectors:

1. Wipe the connector surfaces as shown by the arrows in the figures below with a swab dipped in alcohol.



Type N connectors



2.4 mm, 2.92 mm, 3.5 mm connectors

2. Use compressed air to clean another internal connector surface.
3. Let the alcohol dry on the connector surfaces.
4. Visually inspect the connectors to make sure that no particles or residue remain.
5. Repeat the cleaning procedure if necessary.

WARNING

NEVER use metal items for cleaning connectors.

NEVER wipe the center conductors of female connectors. They should be blown with compressed air.

Gauging Connectors

Gauge the connectors before using the Module for the first time, and regularly during operation.

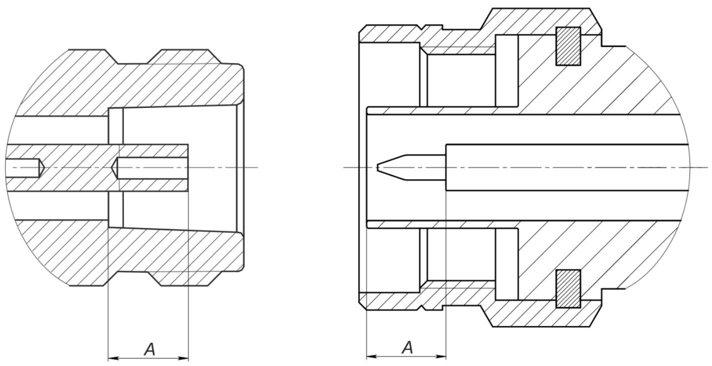
The first gauging of connectors obtains pin depth, which can be used during the Module operation to evaluate its changing.

Gauge the connectors again if:

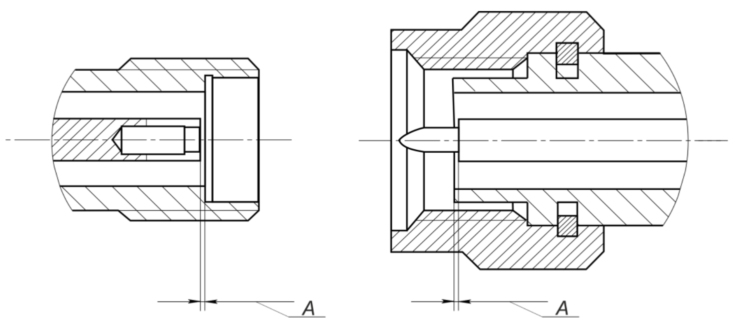
- A visual inspection or Module calibration results suggest that the connector may have defects or damages.
- The device connectors used with the Module are damaged or their pin depth values are out of range for this type of connectors.
- After every 100 connections.

Use gauges for coaxial connectors in compliance with their operating instructions or use multi-purpose tools for linear measurements (for example, micrometer, dial indicator, etc.) to gauge the connectors.

The pin depth of the connectors “PORT A”, “PORT B” and, if available, “PORT C” and “PORT D” are subject to verification. Only measure the A pin depth of type N connectors and 3.5 mm connectors (See figures below).



Type N connectors (female and male)



2.4 mm, 2.92 mm, 3.5 mm connectors (female and male)

The A pin depth value of Module ports connectors must be within the following ranges:

Connectors type	Pin depth range
Type N, female	5.18 to 5.26 mm
Type N, male	5.28 to 5.36 mm
2.4 mm, 2.92 mm, 3.5 mm, male	- 0.08 to 0.00 mm
2.4 mm, 2.92 mm, 3.5 mm, female	- 0.08 to 0.00 mm

The A pin depth value ranges for connectors of other devices are be indicated in their operating manuals.

WARNING

If the pin depth values of the gauged connectors are out of the specified range, such connectors are subject to repair (See Routine Repairs). A device with such connectors is discarded.

Connecting and Disconnecting Devices

The Module connectors should be connected in the following order:

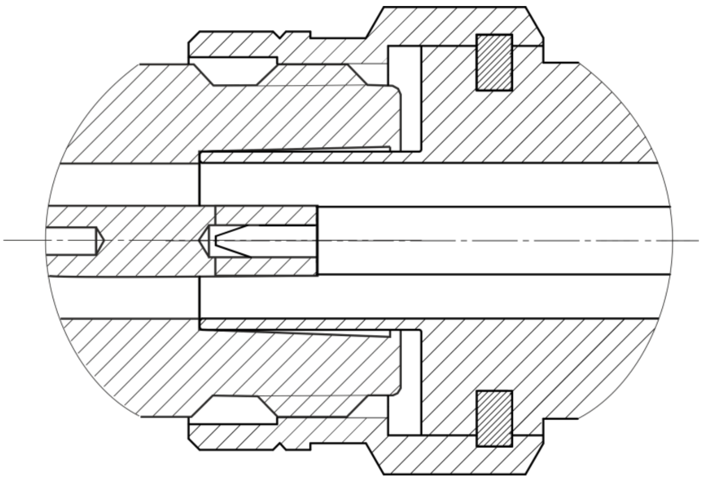
1. Fix the housing of one of the devices being connected. This is necessary to avoid its displacement during connection. Fix the device by any of the following ways:

    - By clamps or wrenches.
    - By weight or configuration of the device itself.
    - By holding the device by hand

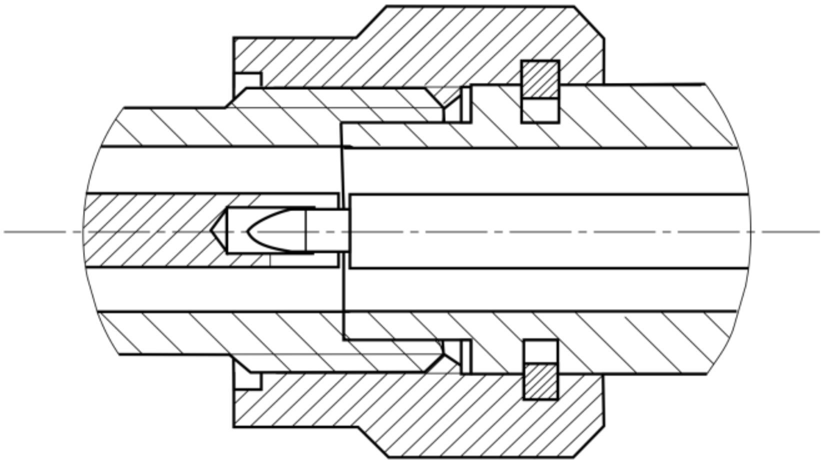
2. Carefully align the connectors of the connected devices.

3. While holding the device being connected, tighten the male connector nut finger tight. Mating plane surfaces of center conductors and outer conductors have to make uniform light contact as shown in the figures below.

4. Tighten the male connector nut using the appropriate torque wrench (the torque value depends on the connector type), while holding the device being connected manually or by using an open-end wrench to keep it from turning. Finally, tighten the male connector nut by holding the wrench at the end of the handle. Tighten the connection just to the torque wrench break point.



Type N connectors (female on the left, male on the right)



2.4 mm, 2.92 mm, 3.5 mm connectors (female on the left, male on the right)

Disconnect the connectors in the following order:

1. Using the torque wrench, which was used for tightening, loosen the male connector nut, while holding the device by hand or an open-end wrench to prevent it from turning.
2. While holding the device so that the connector’s center conductor was at the same straight line as it was connected, turn the male connector nut. Pull the connectors straight apart.

WARNING Do not use alcohol, alkali, or acid for cleaning.

Cleaning and Care of the Protective Housing

The protective housing is not intended for use in extreme environments. Do not bend or stretch the protective housing during use.

Clean the protective housing with a lint-free cloth, slightly dampened with water. Clean the protective housing when it is disassembled.

WARNING	Do not use alcohol, alkali, or acid for cleaning.

Ambient Conditions Control

The measurement accuracy can be severely affected by the change of environmental conditions (especially ambient temperature) between the VNA calibration and the DUT measurements.

The measurements should be performed at an ambient temperature within ±1 °C of the temperature at the time VNA calibration.

Verification

Copper Mountain Technologies recommends following the industry's best practices and user quality policies to determine the ACM verification period. Consider frequency of use, environmental conditions, and storage procedures. The suggested verification interval is 1-3 years.

Routine Repairs

Only authorized routine repair or repair by the licensed company is permitted. The repair method is non-differential.

Routine repairs	Repairs performed to enable or restore the device performance, which includes replacement and/or recovery of separate parts.
Non-differential method	The method of repairs at which the restored constituent parts do not belong to the specific device instance.

Storage Instructions

Module can be stored in the factory packaging at -50 to +70 ºC (-58 °F to 158 °F), a relative humidity of 90% at 25 ºC (77 °F). After the Module has been removed from the factory packaging and while being used, it should be stored at a temperature from+5 ºC to+40 ºC and relative humidity up to 90% at 25 ºC (1 °F to 104 °F).

Keep the storage facilities free from dust, fumes of acids and alkalis, aggressive gases, and other chemicals, which can cause corrosion.

Transportation

Load and unload the Module packages carefully, avoiding shock and packaging damage. Use the markings on the package to place the Modules correctly during transportation.

The Modules must be shipped in any closed vehicle at temperature from -50 to +70 ºC (-58 °F to 158 °F), a relative humidity of 90% at 25 ºC (77 °F).

The Modules can be shipped in packages in conditions excluding any exposure to mechanical or package damage during transportation.

Cargo holds, railway cars, containers, and truck beds, utilized for shipment of the Module should be free from any traces of cement, coal, chemicals, etc. When shipped by air, the products should be kept in aircraft sealed compartments.

Instruction for Use of the Protective Housing

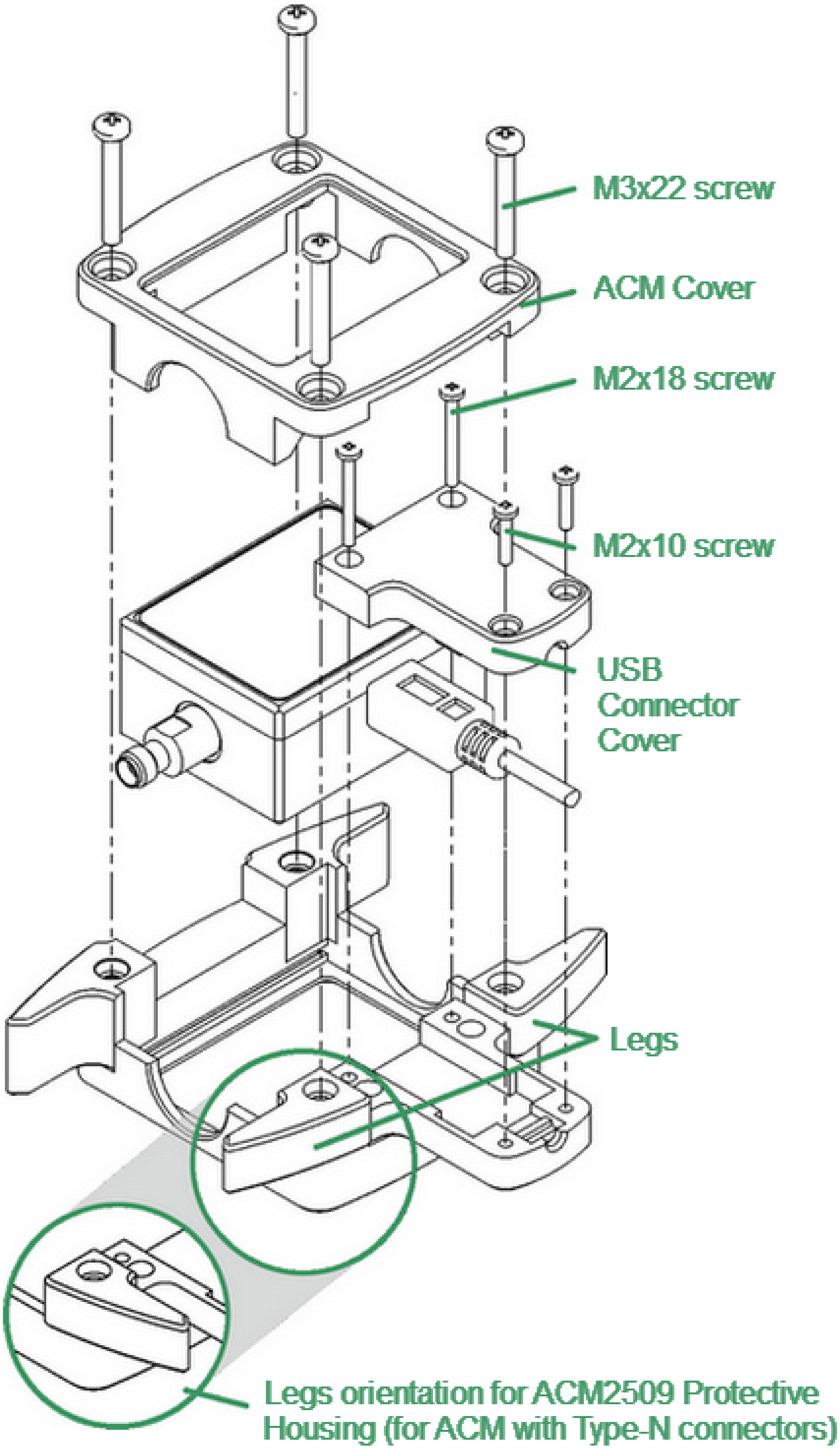
Procedure for installing (removing) the protective housing:

1. Unscrew using a PH1(PZ1) screwdriver:
   - 4 pcs. M3×22 screws on the ACM cover. Remove the ACM cover (See figure below).
   - 2 pcs. M2×18 screws and 2pcs. M2×10 screws on the USB connector cover. Remove the cover.
2. Install (remove) the ACM with the USB cable plugged in. The USB cable must be disconnected from the computer. The orientation of the instrument and the legs of the housing must comply with the figure below.

NOTE	For the ACM2509, turn the legs over for convenient wrench access to the Type-N connectors.

3. Install the USB connector cover, then the ACM cover, using the same screws.

NOTE	The head of the screw should be slightly recessed. Tighten without using force, making sure not to allow the material to bulge on opposite side.



Example of Housing Installation (for ACM2509-011)

Connector Care

When working at frequencies above a few tens of megahertz, the quality and reliability of connections should be monitored more carefully than at lower frequencies. At radio frequencies (RF) and above, the integrity of the transmission line must be maintained throughout the connection, which highlights the importance of the mechanical and electrical compatibility of the connectors.

RF connectors are designed to join devices together as seamlessly as possible. To mate properly, the outer conductor mating surfaces must be clean and flat, and the inner conductor surfaces should come very close together. Even perfectly clean, unused connectors can cause trouble if they are mechanically outside the scope of the specifications. Using a connector gauge is essential, since the critical tolerance in precision microwave connectors is very small.

CAUTION	Damaged or dirty connectors can significantly degrade measurements. To continue to get the best performance from equipment and extend the life of the connectors, perform regular inspections, gauge mechanical tolerances, and clean the RF connectors.

CAUTION	A damaged or out-of-spec connector can destroy the other good connector in just one connection. No device should be used if the connectors are found to be out of the specification.

This document contains operating and maintenance instructions for RF connectors:

- Handling and Storage
- Cleaning
- Gauging
- Connecting and Disconnecting

NOTE	Explore this document and the documentation for gauging before beginning operation with RF connectors.

Handling and Storage

Connectors need to be handled carefully. They should be stored in a safe environment. Always install protective plastic end caps on the connectors of the device when they are not in use.

Keep connectors clean (see Cleaning). Avoid touching the connector mating surfaces with your fingers. Use gloves when working with the connectors to avoid contamination from dirt or grease and to improve accuracy of measurement.

CAUTION	Do not touch mating plane surfaces. Grease and microscopic dirt particles are difficult to remove from these surfaces.

Inspect connectors before mating using a magnifying glass. Check for scratches on the plating, worn mating surfaces, metal particles in the threads or on the mating surfaces, and bent or misaligned conductor centers.

CAUTION	No device should be used if the center connector conductor is bent or broken. No device should be used if the connector has deformed threads.

Holding the connector in your hand or cleaning the connector with compressed air can significantly change its temperature. Wait for the connector temperature to stabilize before using it for calibration or measurement.

Wear a grounding wrist strap and cover the working table with a grounded, conductive mat. This helps to protect devices from electrostatic discharge (ESD).

Connector lifetime:

- All connectors have a limited lifetime. This means that connectors can become defective due to wear during normal use. For best results, all connectors should be inspected and maintained to maximize their lifetime.
- A visual inspection should be performed each time the connectors are mated. Metal particles from connector threads often find their way onto the mating surface during connection or disconnection.

Cleaning

Cleaning off any contamination on the connector mating plane surfaces and threads can extend the lifetime of the connector and improve the quality of calibration and measurement.

Remove loose particles from threads and mating surfaces of the connectors with low-pressure air or nitrogen. Using a compressor is not recommended (air filtration is required), it is safer to use a can. Compressed air is the safest method for cleaning connectors with air dielectrics. Wear safety glasses when cleaning.

If further cleaning is required, a lint-free cleaning swab can be moistened with isopropyl alcohol and applied lightly. If desired, you may clean the connector with a dry cleaning swab without alcohol first. If contamination is still present, use alcohol. Use minimum amount of alcohol.

Only clean connectors with alcohol when there is no power cord connected, ensuring that the instrument is in a well-ventilated area. Allow all residual alcohol moisture to evaporate, and the fumes to dissipate prior to powering up the instrument.

If the connector is still contaminated, use a very small toothpick with a small amount of alcohol applied. Use a magnifying glass when using a toothpick to clean, and apply extreme care to avoid damaging the connector.

CAUTION	Never use any metal objects or any abrasives to clean the connectors. Never use high pressure air (>60 psi). Never allow alcohol into connector support beads. If alcohol unintentionally enters connector support beads, allow the connector to dry for at least 8 hours.

Avoid using too much pressure on the center conductor, as swab fibers can become tangled in the center of the female conductor. When the alcohol evaporates, use compressed air to ensure that the surface is clean.

CAUTION	Never apply lateral force to the center conductor. Never wipe the center conductors of the female connectors. They should be cleaned with compressed air.

Connector cleaning should be performed as follows:

1. Wipe the connector surfaces with the swab moistened with alcohol as shown by arrows (See figures below).

2. Use compressed air to clean the other internal connector surfaces.

3. Let the alcohol evaporate.

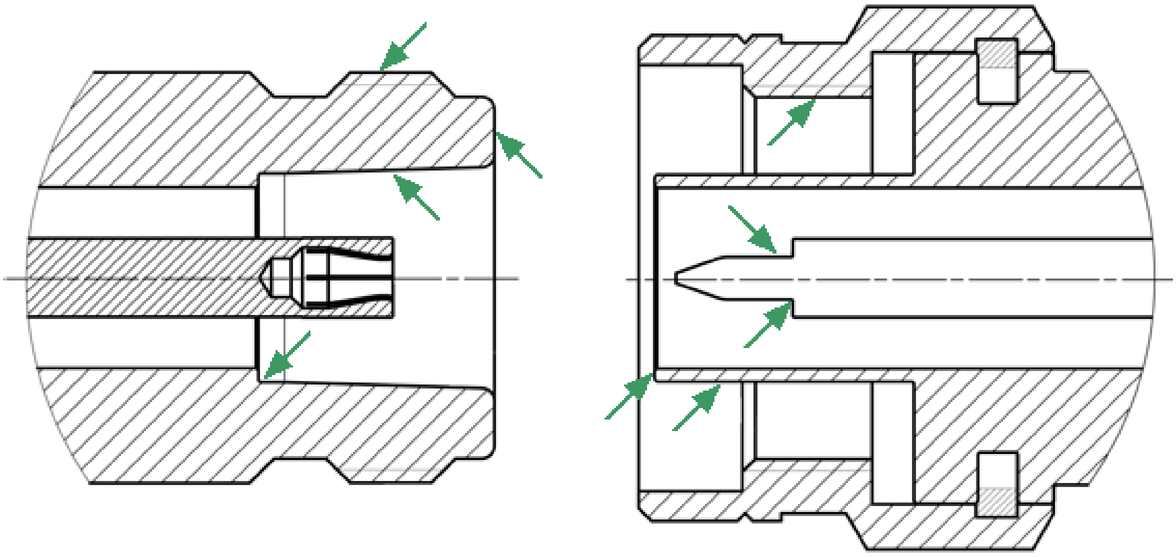
4. Visually inspect the connectors to make sure that no particles or residue remain.

5. Repeat the cleaning procedure if necessary.

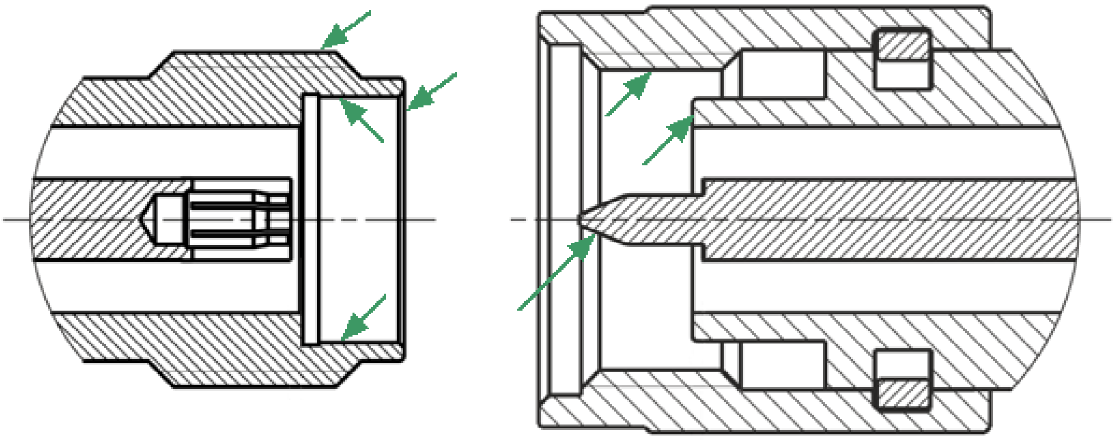
6. If cleaning does not correct any issues, the connector should not be used for measurements.

When cleaning connectors:

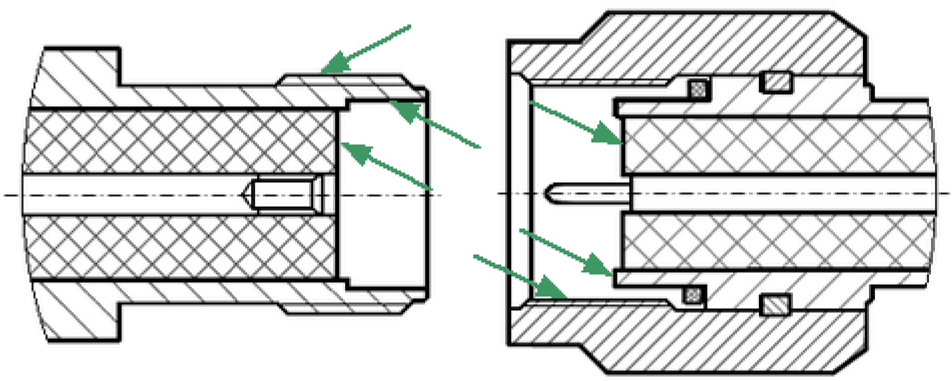
- Always use protective eyewear when using compressed air or nitrogen.
- Keep isopropyl alcohol away from heat, sparks, and flame. Use with adequate ventilation. Avoid contact with eyes, skin, and clothing.
- Avoid electrostatic discharge (ESD). Wear a grounding wrist strap (with a 1 MOhm series resistor) when cleaning connectors.



Type-N Connectors (Female and Male)



3.5 mm NMD, 2.92 mm NMD, 2.4 mm NMD Connectors (Female and Male)



SMA Connectors (Female and Male)

Procedure for Cleaning Connectors

Gauging

Gauging connectors not only provides assurance of proper mechanical tolerances, and thus connector performance, but also indicates when there is potential for causing damage to another connector.

Connector gauging should be performed before the instrument is first used, and during regular operation.

The first gauging of connectors obtains the pin depth, which can be used during operation with the module to evaluate its changes.

Gauge the connectors if:

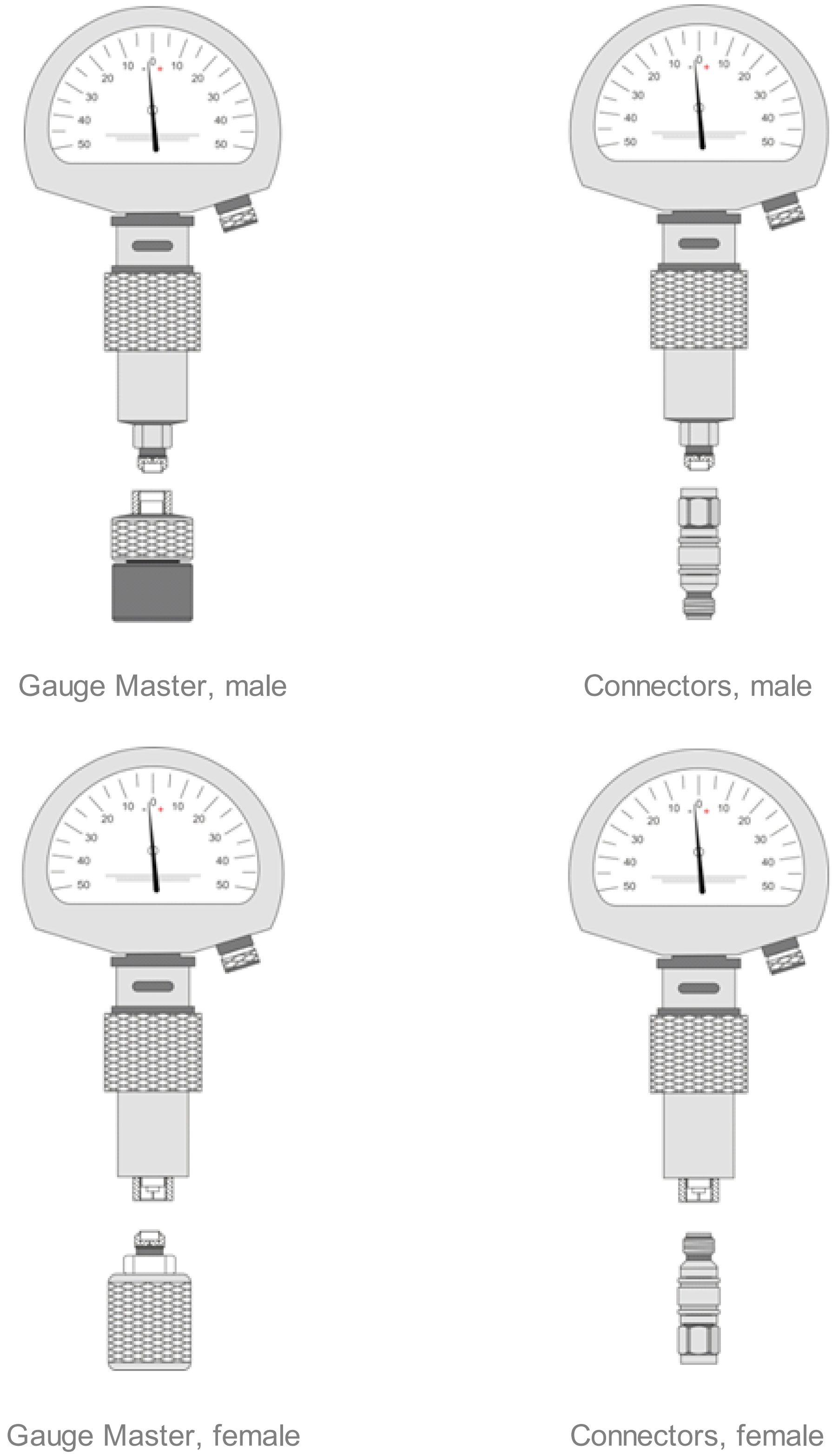
- the device (instrument, calibration standard, cable, adapter, attenuator, or other RF item with coaxial connectors) is being used for the first time.
- visual inspection of the <%DEVICE_NAME%> calibration suggests that the connector may have defects or damage.
- the connectors of the device used with the <%DEVICE_NAME%> are damaged, or their pin depth values are out of the range for this type of connector.
- the device is shared with someone else.
- after every 100 connections or as often as experience suggests.

The procedure for connector gauging is as follows (See figure):

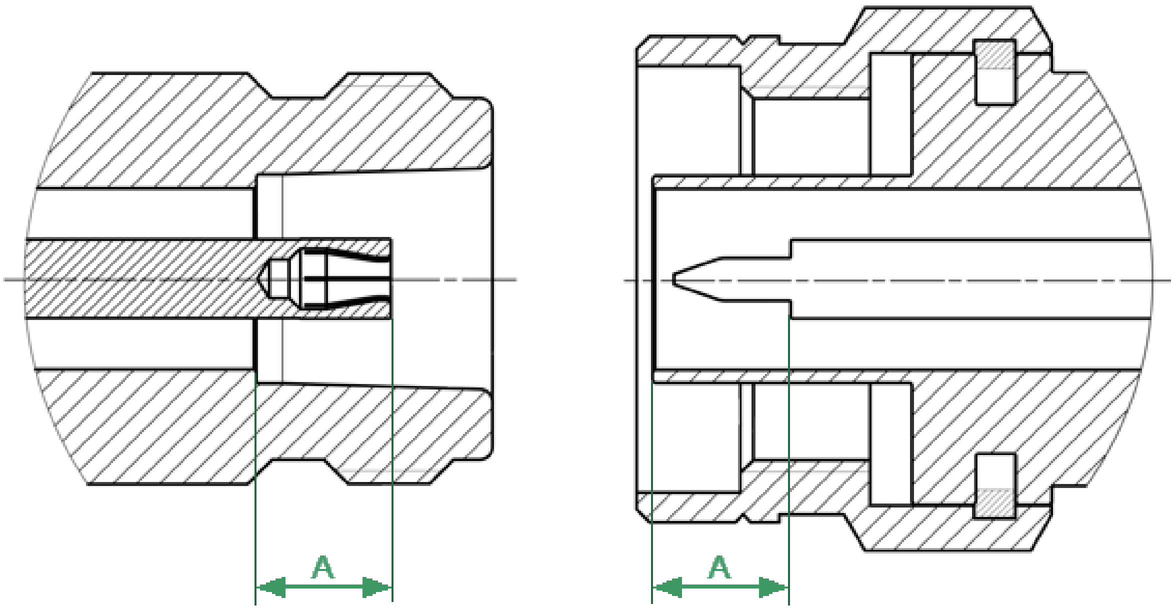
1. Select the proper gauge for your connector.
2. Inspect and clean the gauge, the gauge master, and the connectors to be gauged.
3. Zero the connector gauge before use (according to the gauge documentation).
4. Gauge the connector: while holding the gauge by the barrel, carefully connect the connector under test to the gauge. Read the gauge indicator dial value to determine recession or protrusion and compare the readings with the device specifications (See the figure and table below).

NOTE	Use multiple measurements and keep records of readings.

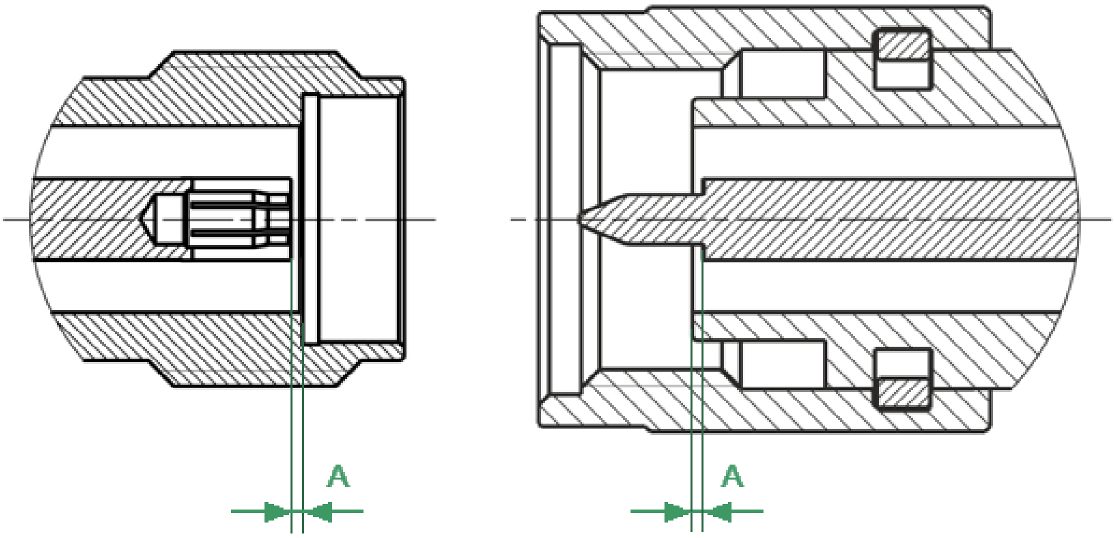
NOTE	Never use an out of specification connector. Do not hold connector gauge by the dial.



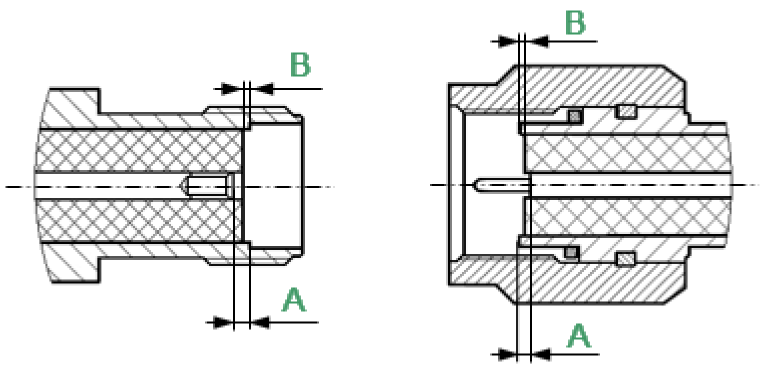
Example of Gauging Connectors



Type-N Connectors (Female and Male)



3.5 mm NMD, 2.92 mm NMD, 2.4 mm NMD Connectors (Female and Male)



SMA Connectors (Female and Male)

Mechanical Requirements for Measured Connectors

The A and B Pin Depth Value of Connector

Connector Type	Pin Acceptable Depth Range	
	A	B
Type-N, female	5.18 to 5.26 mm	–
Type-N, male	5.28 to 5.36 mm	–
3.5 mm NMD, female 2,92 mm NMD, female 2.4 mm NMD, female	-0.08 to 0.00 mm	–
3.5 mm NMD, male 2,92 mm NMD, male 2.4 mm NMD, male	-0.08 to 0.00 mm	–
SMA, female SMA, male	-0.10 to 0.00 mm	-0.05 to 0.00 mm

If the pin depth values of the gauged connectors are out of the acceptable range, the connectors may be eligible to be sent in for repair.

Connecting and Disconnecting

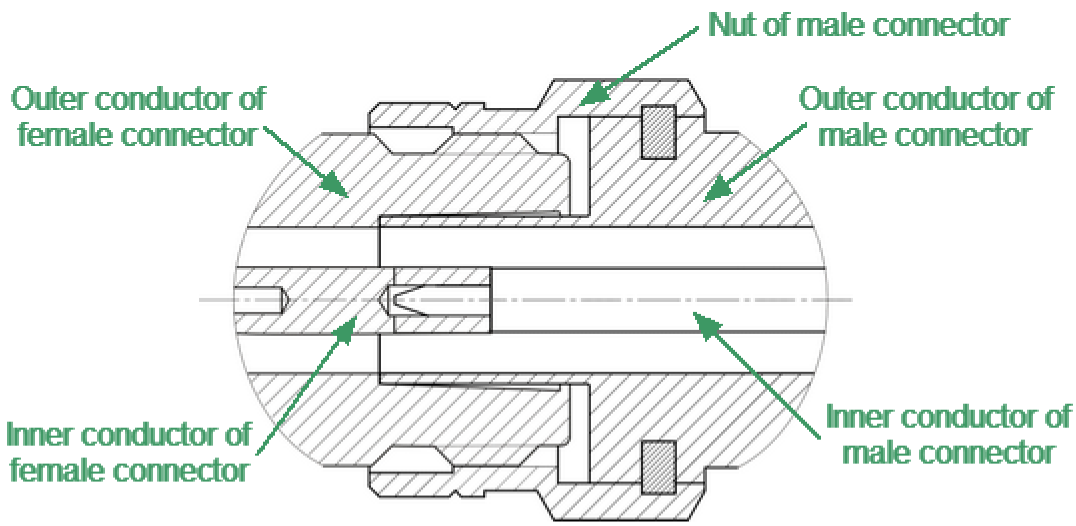
When operating the <%DEVICE_NAME%>, it is often necessary to connect various devices to each other: cables to <%DEVICE_NAME%> measurement ports, junctions to cables, calibration tools to junctions or <%DEVICE_NAME%> ports, devices under test to ports, etc.

Connecting

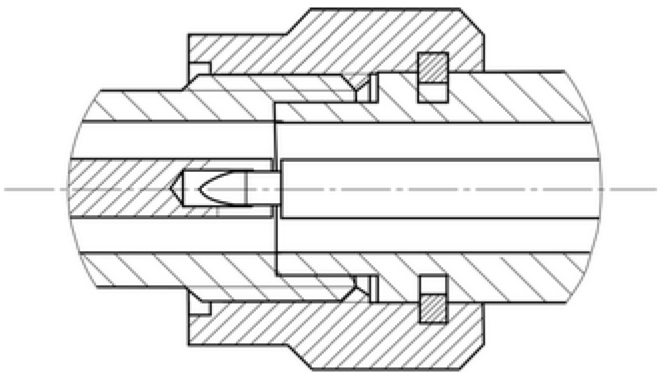
Connect devices with coaxial connectors in the following sequence to ensure maximum repeatability of measurement results, as well as to prevent breakage:

1. Carefully align the connectors of the devices being connected.

2. While holding the device that is being connected, tighten the male connector nut manually. The mating plane surfaces of the center conductors and the outer conductors must make uniform light contact, as shown in figure below.



Type-N Connectors (Female on the Left, and Male on the Right)



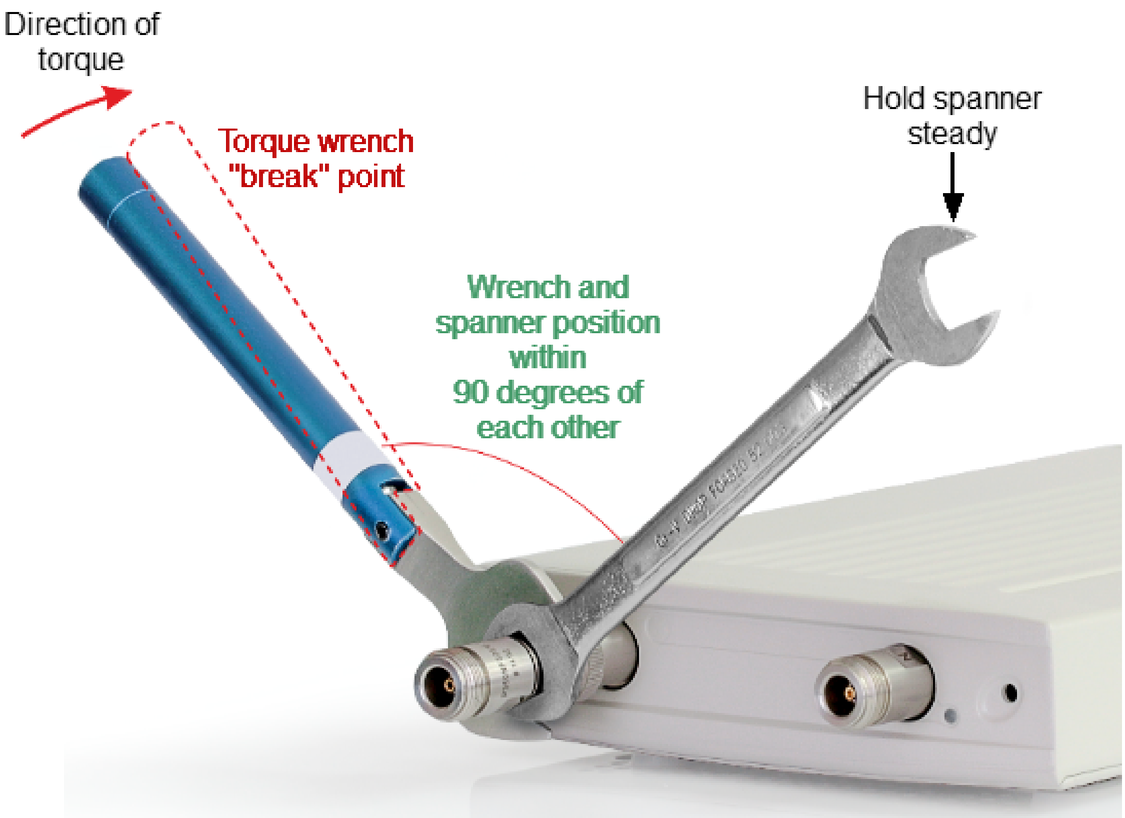
3.5 mm NMD, 2.92 mm NMD, 2.4 mm NMD Connectors (Female on the Left, and Male on the Right)

SMA Connectors (Female on the Left, and Male on the Right)

Connecting Example

3. Tighten the male connector nut using the appropriate torque wrench while holding the device being connected, or hold the device by using an open-end spanner to keep it from rotating. Position the wrench and spanner within 90 degrees of each other before applying force. Finally, tighten the male connector nut by holding the wrench at the end of the handle. Tighten the connection just to the torque wrench “break” point (See figures below).

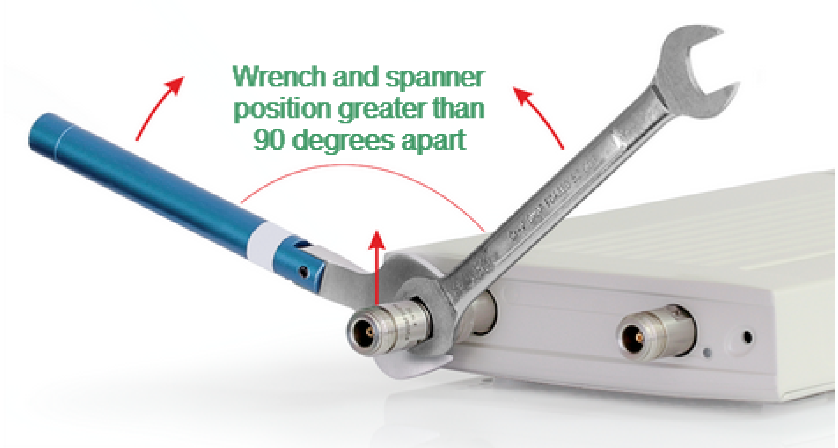
Hold the torque wrench by the end of the handle when tightening. The torque value depends on the connector type (See table below).



Correct Torque Wrench and Spanner Positions

CAUTION

The wrench and spanner should not be positioned more than 90 degrees apart. A larger degree of separation can cause excessive misalignment of the connectors.



Incorrect Usage of Torque Wrench and Spanner (Too Much Lift on Connection).

Recommended Torque Values

Connector Type	Recommended Torque Values
Type-N	1.1 to 1.7 N·m (12 in. lbs)
3.5 mm NMD, 2.92 mm NMD, 2.4 mm NMD	0.8 to 1.0 N·m (8 in. lbs)
SMA	0.56 N·m (5 in. lbs)

CAUTION

The jumper cables will be damaged if more than 0.9 N·m of torque is applied to their SMA connectors.

Do not exceed the permissible torque value.

CAUTION	When making and breaking connections, connector mating surfaces should not rotate. Rotate the nut of the male connector only. Avoid rotating the devices. Use a suitable torque wrench.

CAUTION	Never cross-thread the connection. Never twist the connector body to make the connection. Never mate the connectors of incompatible types.

Disconnecting

Disconnect the connectors in the following order:

1. Using the torque wrench used for tightening, loosen the male connector nut while holding the device, or hold the device with an open-end wrench to prevent it from turning.

2. Turn the male connector nut while holding the device so that the connector center conductor remains in the same straight line position as it was connected. Pull the connectors straight apart.

Glossary

Prefixes

μ	micro (10^{-6})
m	milli (10^{-3})
k	kilo (10^{3})
M	Mega (10^{6})
G	Giga (10^{9})

Number / Symbols

Ω	ohm
dB	decibel
dBm	decibels above 1 milliwatt
W	Watt
F	Farad
H	Henry
Hz	Hertz
m	meter
sec	second
V	Volt

ACM	Automatic Calibration Module
CMT	Copper Mountain Technologies
CW	Continuous Wave
DC	Direct Current
DSP	Digital Signal Processor
DUT	Device Under Test
IF	Intermediate Frequency
LED	Light-emitting diode
LRL	Line-Reflect-Line calibration
PC	Personal Computer
RF	Radio Frequency
SCPI	Standard Commands for Programmable Instruments
S-parameters	Scattering parameters of linear electrical network
SOLT	Short-Open-Load-Through Calibration
SOLR	Short-Open-Load-Reciprocal Calibration
SWR	Standing Wave Ratio
TRL	Thru-Reflect-Line Calibration
USB	Universal Serial Bus
VNA	Vector Network Analyzer