COPPER MOUNTAIN
TECHNOLOGIES

1-Port VNA Series

R54

R140

R60

R180/RP180

Operating Manual

Software version 19.1.0

March 2019

T A B L E O F C O N T E N T S

INTRODUCTION

This Operating Manual represents design, specifications, overview of functions, and detailed operation procedure for the Vector Network Analyzer, to ensure effective and safe use of the technical capabilities of the instrument by the user.

Vector Network Analyzer operation and maintenance should be performed by qualified engineers with initial experience in operating of microwave circuits and PC.

The following abbreviations are used in this Manual:

PC – Personal Computer

DUT – Device Under Test

IF – Intermediate Frequency

CW – Continuous Wave

SWR – Standing Wave Ratio

VNA – Vector Network Analyzer

SAFETY INSTRUCTIONS

Carefully read through the following safety instructions before putting the Analyzer into operation. Observe all the precautions and warnings provided in this Manual for all the phases of operation, service, and repair of the Analyzer.

The VNA must be used only by skilled and specialized staff or thoroughly trained personnel with the required skills and knowledge of safety precautions.

The Analyzer complies with INSTALLATION CATEGORY I as well as POLLUTION DEGREE 2 in IEC61010−1.

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

The Analyzer is tested in stand-alone condition or in combination with the accessories supplied by Copper Mountain Technologies against the requirement of the standards described in the Declaration of Conformity. If it is used as a system component, compliance of related regulations and safety requirements are to be confirmed by the builder of the system.

Never operate the Analyzer in the environment containing inflammable gasses or fumes.

Operators must not remove the cover or 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 your Analyzer when connected or disconnected from the DUT. Static charge can build up on your body and damage the sensitive circuits of 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 PC and the body of the DUT to protective grounding before you start operation.

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.

1. GENERAL OVERVIEW

1.1 Description

The VNA is designed for use in the process of development, adjustment and testing of antenna-feeder devices in industrial and laboratory facilities, as well as in field, including operation as a component of an automated measurement system. The Analyzer is designed for operation with external PC, which is not supplied with it.

1.2 Specifications

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

1.3 Measurement Capabilities

Measured parameters	S_{11}, Cable loss.
Number of measurement channels	Up to 4 logical channels. Each logical channel is represented on the screen as an individual channel window. A logical channel is defined by such stimulus signal settings as frequency range, number of test points, etc.
Data traces	Up to 4 data traces can be displayed in each channel window. A data trace represents one of such parameters of the DUT as magnitude and phase of S_{11}, DTF, Cable loss.
Memory traces	Each of the 4 data traces can be saved into memory for further comparison with the current values.
Data display formats	SWR, Return loss, Cable loss, Phase, Expanded phase, Smith chart diagram, DTF SWR, DTF Return loss, Group delay.
Sweep setup features	
Sweep type	Linear frequency sweep, logarithmic frequency sweep, and segment frequency sweep.
Measured points per sweep	Set by user from 2 to 100,001.

Segment sweep	A frequency sweep within several user-defined segments. Frequency range, number of sweep points, IF bandwidth and measurement delay should be set for each segment.
Power settings	Two modes of output power level. Power levels depending on device.
Sweep trigger	Trigger modes: continuous, single, hold.
	Trace display functions
Trace type	Data trace, memory trace.
Trace math	Data trace modification by math operations: addition, subtraction, multiplication or division of measured complex values and memory data.
Auto scaling	Automatic selection of scale division and reference level value to have the trace most effectively displayed.
Electrical delay	Calibration plane moving to compensate for the delay in the test setup. Compensation for electrical delay in a DUT during measurements of deviation from linear phase.
Phase offset	Phase offset defined in degrees.
	Accuracy enhancement
Calibration	Calibration of a test setup (which includes the Analyzer and adapter) significantly increases the accuracy of measurements. Calibration allows to correct the errors caused by imperfections in the measurement system: system directivity, source match, and tracking.
Calibration methods	The following calibration methods are available: • reflection normalization; • full one-port calibration.
Reflection normalization	The simplest calibration method.
Full one-port calibration	Method of calibration that ensures high accuracy.

Factory calibration	The factory calibration of the Analyzer allows performing measurements without additional calibration and reduces the measurement error after reflection normalization.
Mechanical calibration kits	The user can select one of the predefined calibration kits of various manufacturers or define own calibration kits.
Electronic calibration modules	Electronic calibration modules manufactured by COPPER MOUNTAIN TECHNOLOGIES make the Analyzer calibration faster and easier than traditional mechanical calibration.
Defining of calibration standards	Different methods of calibration standard defining are available: • standard defining by polynomial model; • standard defining by data (S-parameters).
Error correction interpolation	When the user changes such settings as start/stop frequencies and number of sweep points, compared to the settings of calibration, interpolation or extrapolation of the calibration coefficients will be applied.
	Marker functions
Data markers	Up to 16 markers for each trace. A marker indicates stimulus value and the measured value in a given point of the trace.
Reference marker	Enables indication of any maker values as relative to the reference marker.
Marker search	Search for max, min, peak, or target values on a trace.
Marker search additional features	User-definable search range. Functions of specific condition tracking or single operation search.
Setting parameters by markers	Setting of start, stop and center frequencies by the stimulus value of the marker and setting of reference level by the response value of the marker.
Marker math functions	Statistics, bandwidth, flatness, RF filter.

Statistics	Calculation and display of mean, standard deviation and peak-to-peak in a frequency range limited by two markers on a 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	The function of conversion of the S-parameters measured at 50 Ω port into the values, which could be determined if measured at a test port with arbitrary impedance.
De-embedding	The function allows to exclude mathematically the effect of the fixture circuit connected between the calibration plane and the DUT from the measurement result. This circuit should be described by an S-parameter matrix in a Touchstone file.
Embedding	The function allows to simulate mathematically the DUT parameters after virtual integration 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	The 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	The function performs data transformation from frequency domain into response of the DUT to radiopulse in time domain. Time domain span is set by the user arbitrarily from zero to maximum, which is determined by the frequency step. Windows of various forms allow better tradeoff between resolution and level of spurious sidelobes.
Time domain gating	The function mathematically removes unwanted responses in time domain what allows obtaining frequency response without influence from the fixture elements. The function applies reverse transformation back to frequency domain from the user-defined span in time domain. Gating filter types are: bandpass or notch. For better tradeoff between gate resolution and level of spurious sidelobes the following filter shapes are available: maximum, wide, normal and minimum.
Other features	
Analyzer control	Using external personal computer via USB interface.
Familiar graphical user interface	Graphical user interface based on Windows operating system ensures fast and easy Analyzer operation by the user. The software interface of Analyzers is compatible with modern tablet PCs and laptops.
Saving trace data	Saving the traces in graphical format and saving the data in Touchstone and *.csv (comma separated values) formats on the hard drive are available.
Remote control	
COM/DCOM	Remote control via COM/DCOM. COM automation runs the user program on an Analyzer PC. DCOM automation runs the user program on a LAN-networked PC. Automation of the instrument can be achieved in any COM/DCOM-compatible language or environment, including Python, C++, C#, VB.NET, LabVIEW, MATLAB, Octave, VEE, Visual Basic (Excel) and many others.
Socket	Data transfer between the PC user and the computer that is connected to the device, can be also performed via Socket (TCP, port 5025).

1.4 Principle of Operation

The Analyzer consists of the Analyzer Unit, some supplementary accessories, and personal computer (which is not supplied with the package). The Analyzer Unit is powered and controlled by PC via USB-interface. The block diagram of the Analyzer is represented in Figure 1.1.

The Analyzer Unit consists of a source oscillator, a local oscillator, a source power attenuator, a directional coupler and other components which ensure the Analyzer operation. The test port is the source of the test signal. The incident and reflected signals from the directional coupler are supplied into the mixers, where they are converted into IF, and are transferred further to the 2-channel receiver. The 2-channel receiver, after filtration, digitally encodes the signals and supplies them for further processing (filtration, phase difference measurement, magnitude measurement) into the signal processor. The filters for the IF are digital and have passband from 10 Hz to 30(100) kHz. The combination of the assemblies of directional couplers, mixers, and 2-channel receiver forms two similar signal receivers.

An external PC controls the operation of the components of the Analyzer. To fulfill the S-parameter measurement, the Analyzer supplies the source signal of the assigned frequency from test port to the DUT, then measures magnitude and phase of the signal reflected by the DUT, and after that compares these results to the magnitude and phase of the source signal.

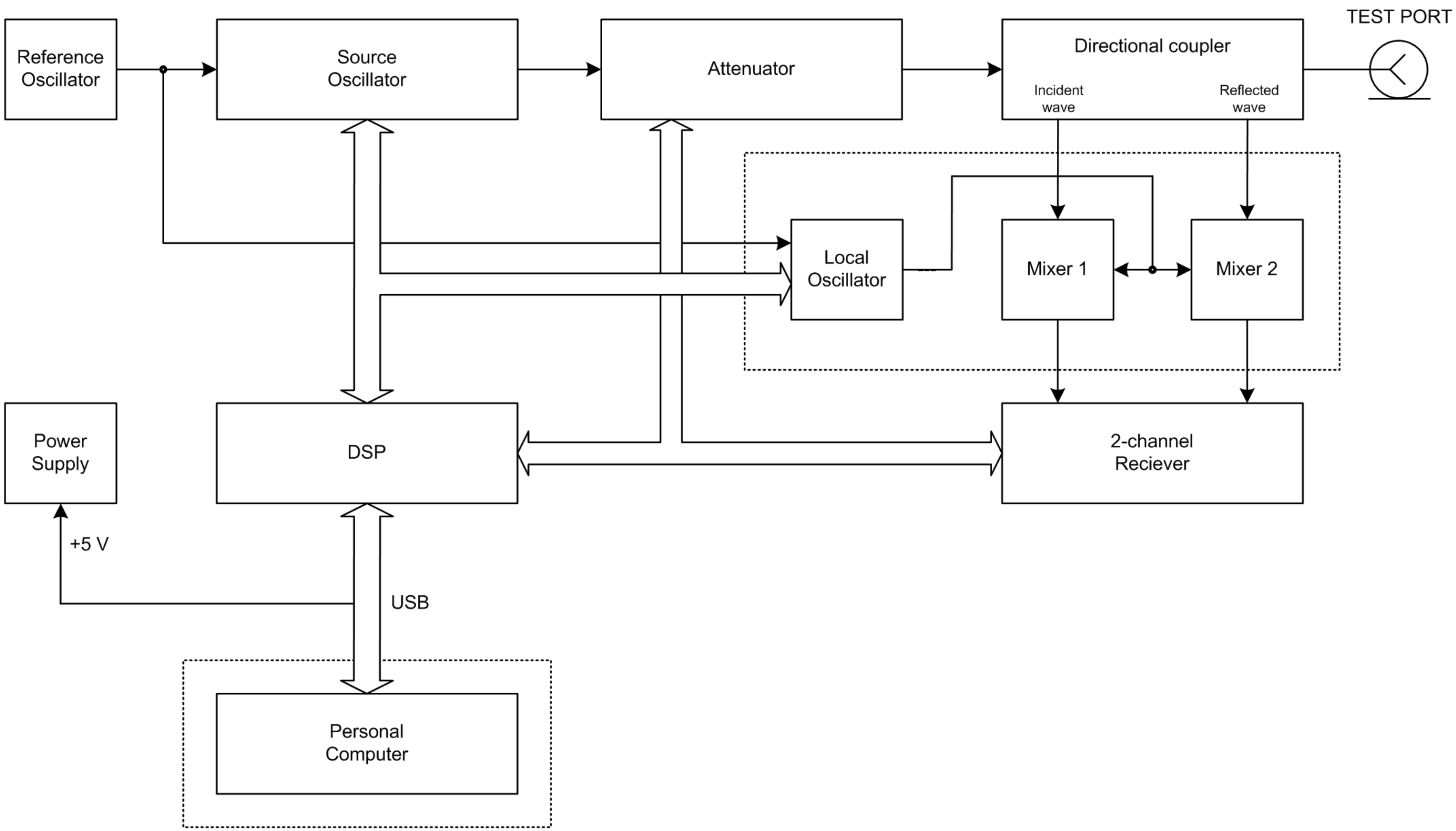


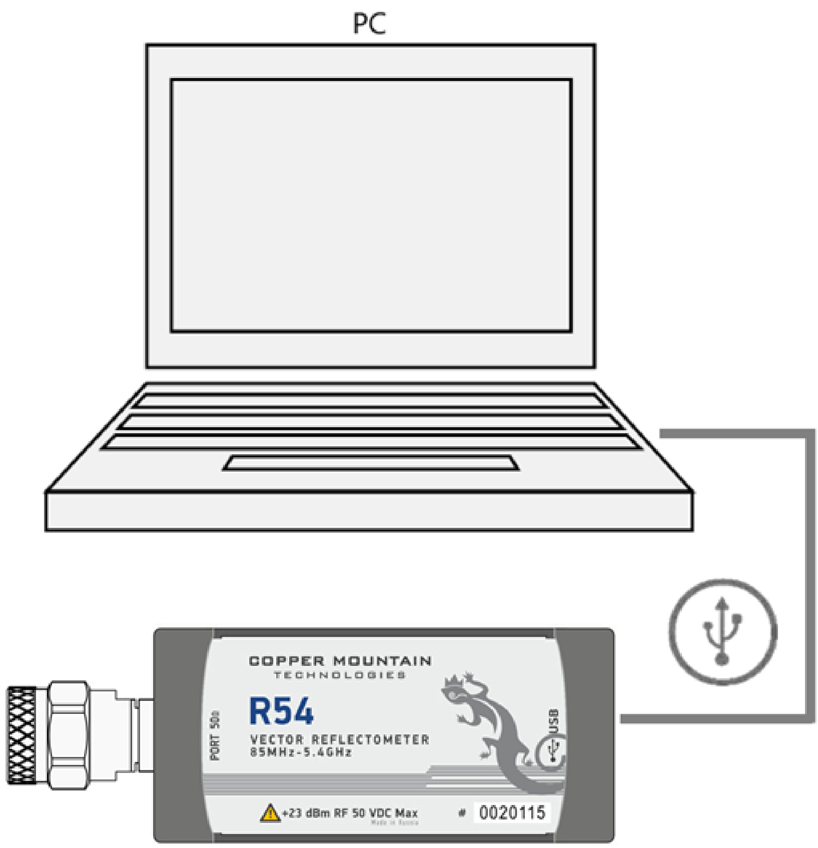
Figure 1.1The VNA block diagram

2. PREPARATION FOR USE

2.1 General Information

Unpack the VNA and other accessories.

Connect the Analyzer to the PC using the USB Cable supplied in the package. Install the software (supplied on the flash drive) onto your PC. The software installation procedure is described below.

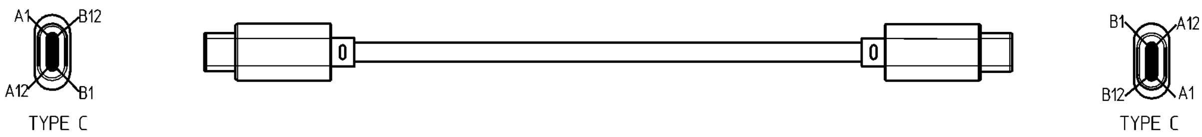


Warm-up the Analyzer for the time stated in its specifications.

Assemble the test setup using cables, connectors, fixtures, etc, which allow DUT connection to the Analyzer.

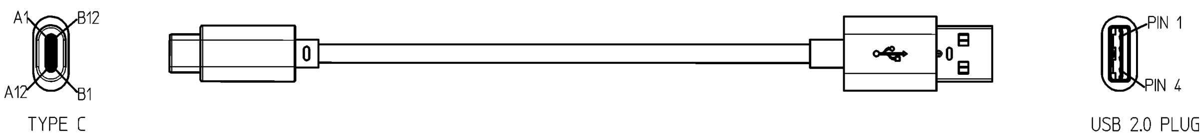
Perform calibration of the Analyzer. Calibration procedure is described in section 5.

Attention!	To avoid motherboard damage you must use USB cables supplied in the package or similar ones according to the specifications shown in Figure 2.1 and Figure 2.2 (for R180/RP180 only)



PIN ASSIGNMENTS				
Type C(P1)	SIGNAL NAME	WIRE COLOR	SIGNAL NAME	Type C(P2)
A1,B1,A12,B12	GND	BLACK	GND	A1,B1,A12,B12
A4,B4,A9,B9	VBUS	RED	VBUS	A4,B4,A9,B9
A5	CC	BLUE	CC	A5
B5	VCONN	-	VCONN	B5
A6	Dp1	GREEN	Dp1	A6
A7	Dn1	WHITE	Dn1	A7
SHELL	SHIELD	BRAID	SHIELD	SHELL

Figure 2.1 USB TYPE C TO C 2.0, 3A



Rp
56Kohm

PIN ASSIGNMENTS				
Type C	SIGNAL NAME	WIRE COLOR	USB 2.0	SIGNAL NAME
A1,B1,A12,B12	GND	BLACK	4	GND
A4,B4,A9,B9	VBUS	RED	1	VBUS
A5	CC	-	-	-
B5	VCONN	-	-	-
A6	Dp1	GREEN	3	D+
A7	Dn1	WHITE	2	D-
SHELL	SHIELD	BRAID	SHELL	SHIELD

Figure 2.2 USB TYPE C TO USB 2.0 A MALE, 3A

2.2 Software Installation

The software is installed to the external PC running under Windows operating system. The Analyzer is connected to the external PC via USB interface.

Minimal system requirements for the PC	WINDOWS 7 and Higher 1.5 GHz Processor 2 GB RAM USB 2.0 High Speed

The supplied USB flash drive contains the following software:

Flash drive contents	Setup_RVNA_vX.X.X.exe installer file (**X.X.X** – program version number); **Driver** folder contains the driver; **Doc** folder contains documentation.

The procedure of the software installation is performed in two steps. The first one is the driver installation. The second step comprises installation of the program, documentation and other related files.

Driver installation	Connect the Analyzer to your PC via the supplied USB cable. When you connect the Analyzer to the PC for the first time, Windows will automatically detect the new USB device and will open the USB driver installation dialog (Windows 7 and higher). In the USB driver installation dialog, click on Browse and specify the path to the driver files, which are contained in the Driver folder on the USB flash drive.
Program and related files installation	Run the Setup_RVNA_vX.X.X.exe installer file from the supplied USB flash drive. Follow the instructions of the installation wizard.

2.3 Top Panel

The top panel view of Analyzers is represented in the figures below. The top panel is equipped with the READY/STANDBY LED indicator running in the following modes:

- green blinking light – standby mode. In this mode the current consumption of the device from the USB port is minimum;
- green glowing light – normal device operation.

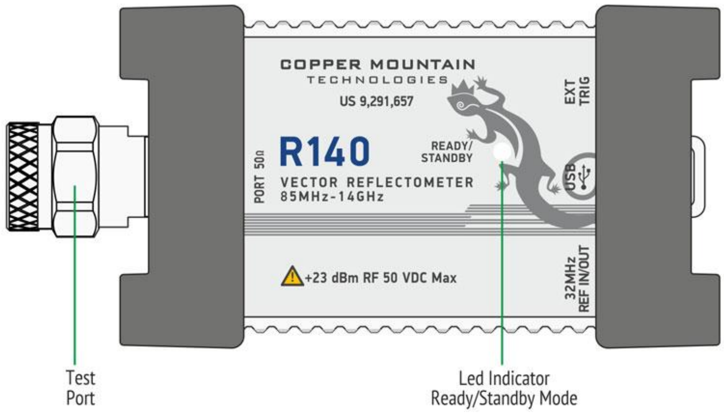


Figure 2.3 R140 top panel

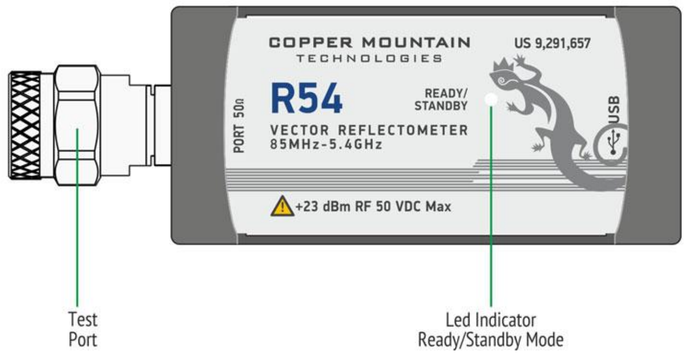


Figure 2.4 R54 top panel

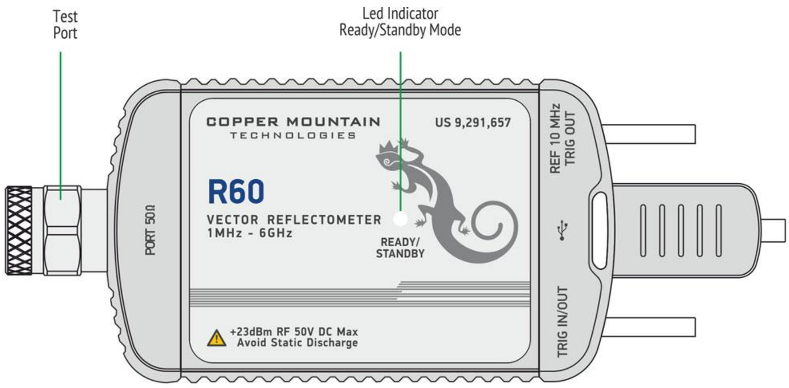


Figure 2.5 R60 top panel

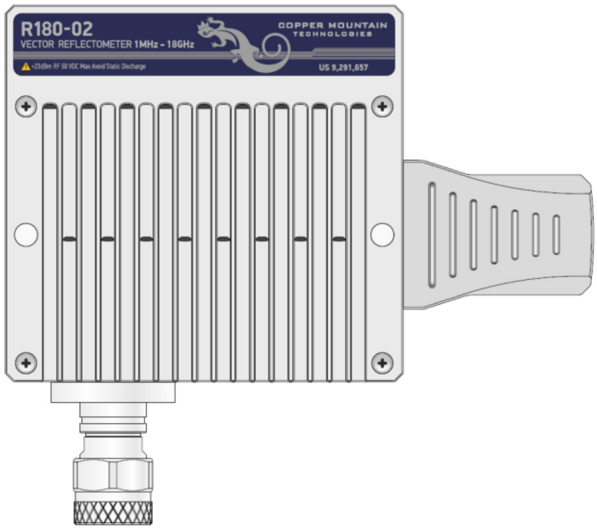


Figure 2.6 R180 top panel

2.4 Test Port

The test port (type-N male 50 Ω) is intended for DUT connection. It is also used as a source of the stimulus signal and as a receiver of the response signal from the DUT.

2.5 Mini B USB Port

The mini B USB port view is represented in Figure 2.7, Figure 2.8, Figure 2.9 and Figure 2.10. It is intended for connection to USB port of the personal computer via the supplied USB cable.

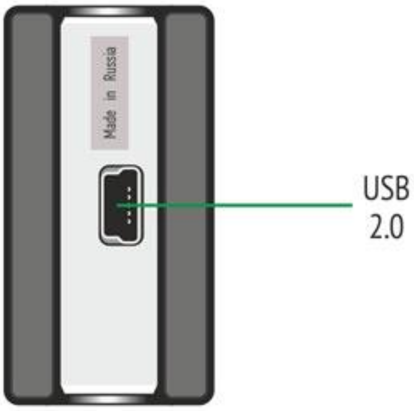


Figure 2.7 Mini B USB port R54

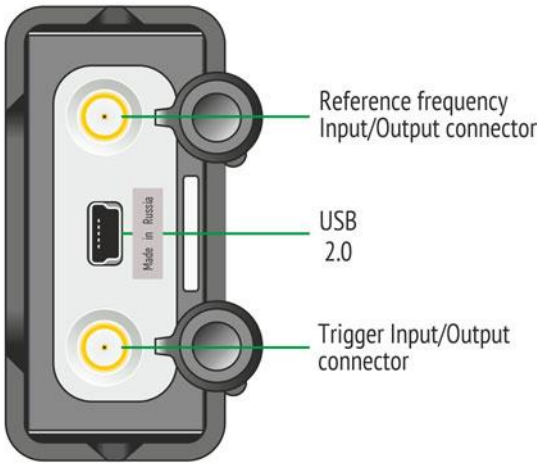


Figure 2.8 Mini B USB port R140

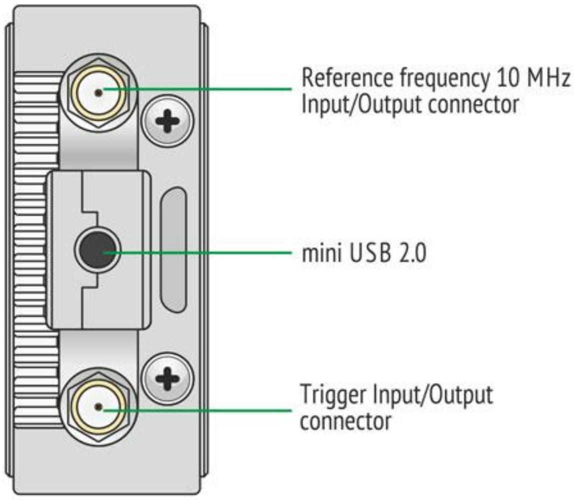


Figure 2.9 Mini B USB port R60

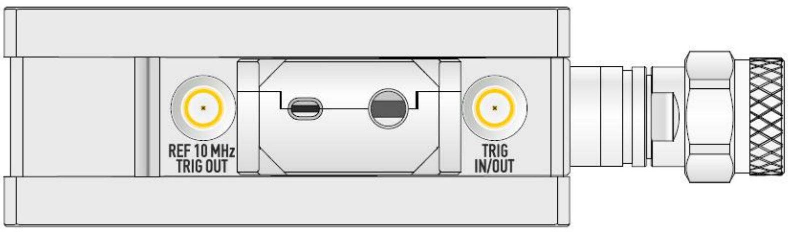


Figure 2.10 Mini B USB port R180

2.6 External Trigger Signal Input Connector (R140 model only)

This connector allows the user to connect an external trigger source. Connector type is SMA female. TTL compatible inputs of 3 V to 5 V magnitude have up to 1 us pulse width. Input impedance is at least 10 kΩ.

2.7 External Reference Frequency Input Connector (R140 model only)

External reference frequency - see in its specifications, input level is 2 dBm ± 2 dB, input impedance at «Ref In» is 50 Ω. Connector type is SMA female.

2.8 Reference Frequency Input/Output Connector (R60 and R180 model only)

External reference frequency is 10 MHz, input level is 2 dBm ± 2 dB, input impedance is 50 Ohm. Output reference signal level is 3 dBm ± 2 dB into 50 Ohm impedance. Connector type is SMA female.

2.9 External Trigger Signal Input/Output Connector (R60 and R180 model only)

External Trigger Signal Input allows the user to connect an external trigger source. Connector type is SMA female. 3.3v CMOS TTL compatible inputs magnitude have at least 1 μs pulse width. Input impedance is at least 10 kOhm. The External Trigger Signal Output port can be used to provide trigger to an external device. The port outputs various waveforms depending on the setting of the Output Trigger Function: before frequency setup pulse, before sampling pulse, after sampling pulse, ready for external trigger, end of sweep pulse, measurement sweep.

3. GETTING STARTED

This section represents a sample session of the Analyzer. It describes the main techniques of measurement of reflection coefficient parameters of the DUT. SWR and reflection coefficient phase of the DUT will be analyzed.

The instrument sends the stimulus to the input of the DUT and then receives the reflected wave. Generally in the process of this measurement the output of the DUT should be terminated with a LOAD standard. The results of these measurements can be represented in various formats. The given example represents the measurement of SWR and reflection coefficient phase.

Typical circuit of DUT reflection coefficient measurement is shown in Figure 3.1.

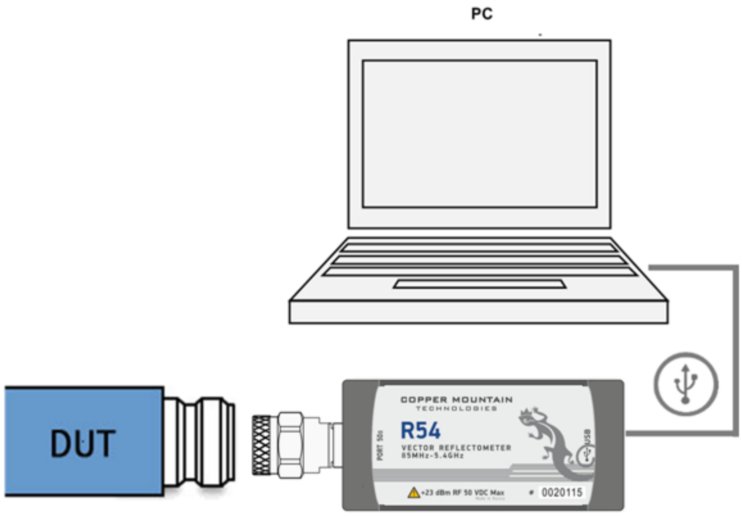


Figure 3.1.

To measure SWR and reflection coefficient phases of the DUT in the given example you should go through the following steps:

- Prepare the Analyzer for reflection measurement;
- Set stimulus parameters (frequency range, number of sweep 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 calibration of the Analyzer for reflection coefficient measurement;
- Analyze SWR and reflection coefficient phase using markers.

3.1 Analyzer Preparation for Reflection Measurement

Turn on the Analyzer and warm it up for the period of time stated in the specifications.

Ready state features	The bottom line of the screen displays the instrument status bar. It should read **Ready**.

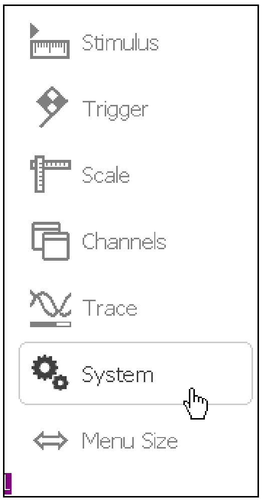
Connect the DUT to the test port of the Analyzer. Use the appropriate adapters for connection of the DUT input to the Analyzer test port. If the DUT input is type-N (female), you can connect the DUT directly to the port.

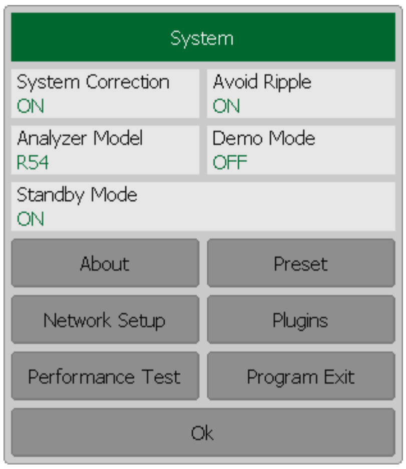
3.2 Analyzer Presetting

Before you start the measurement session, it is recommended to reset the Analyzer into the initial state. The initial condition setting is described in Appendix 1.

Note	You can operate either by the mouse or using a touch screen.

To restore the initial state of the Analyzer use the following softkeys in the right menu bar **System > Preset.**





Close the dialog by **Ok.**

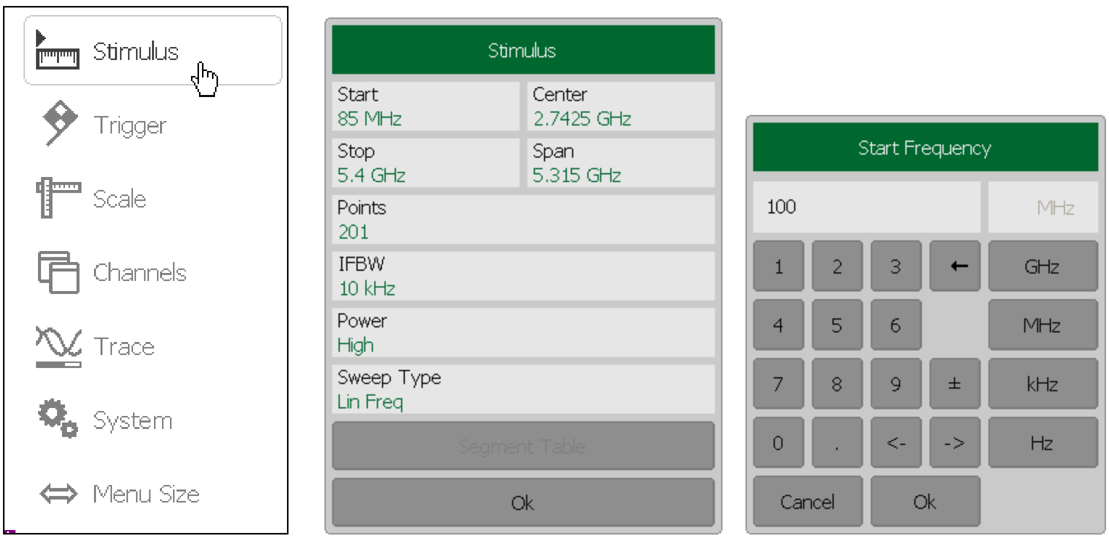
3.3 Stimulus Setting

After you have restored the preset state of the Analyzer, the stimulus parameters will be as follows: full frequency range of the instrument, sweep type is linear, number of sweep points is 201, power level is high, and IF is 10 kHz.

For the current example, set the frequency range from 100 MHz to 1 GHz.

To set the start frequency of the frequency range to 100 MHz use the following softkey in the right menu bar **Stimulus .**

Then select the **Start** field and enter 100 using the on-screen keypad. Complete the setting by **Ok.**

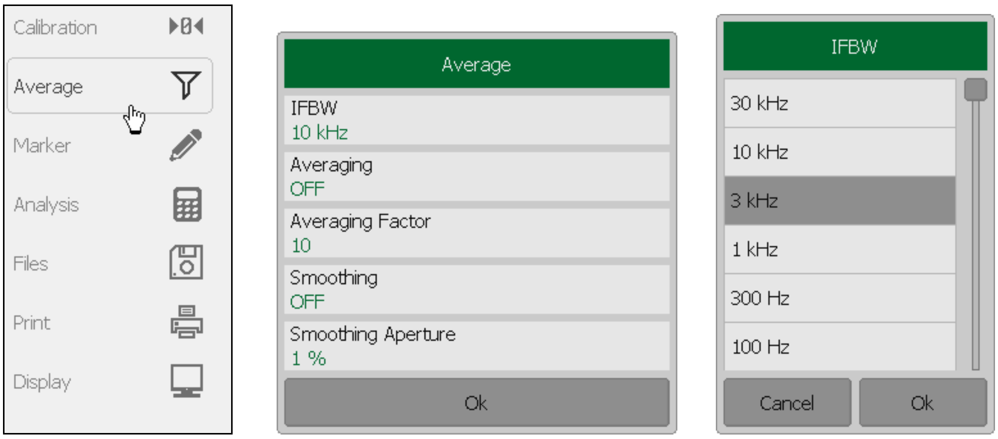


To set the stop frequency of the frequency range to 1 GHz select the **Stop** field and enter 1000 using the on-screen keypad. Complete the setting **Ok.** Close the **Stimulus** dialog by **Ok.**

3.4 IF Bandwidth Setting

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

To set the IF bandwidth to 3 kHz use the following softkey in the left menu bar **Average.**



Then select the **IFBW** field in the **Average** dialog.

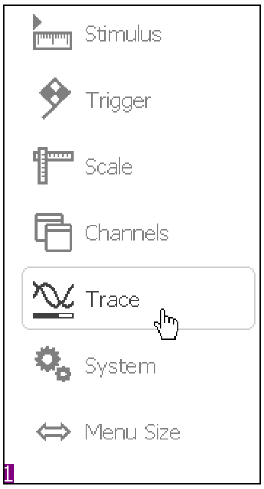
To set the IF bandwidth in the **IFBW** dialog use the following softkeys **3 kHz > Ok.**

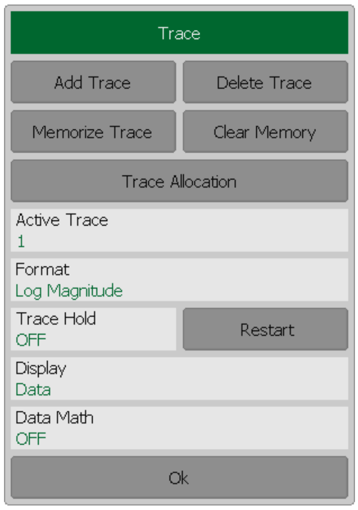
Note	You can also select the IF bandwidth by double clicking on the required value in the **IFBW**. The dialog will close automatically.

3.5 Number of Traces, Measured Parameter and Display Format Setting

In the current example, two traces are used for simultaneous display of the two parameters (SWR and reflection coefficient phase).

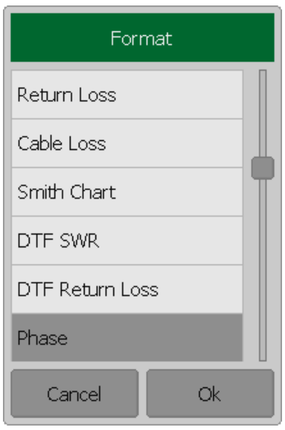
To add the second trace use the following softkeys in the right menu bar **Trace > Add trace.**

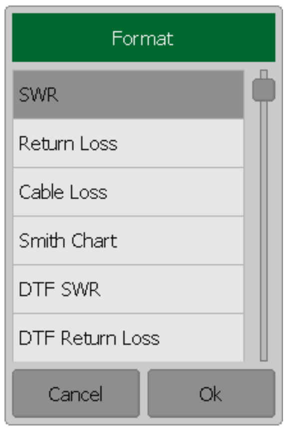




The added trace automatically becomes active. The active trace is highlighted in the list and on the graph.

To select the trace display format click on **Format**.





Set the Phase format by **Phase > Ok.**

To scroll up and down the formats list click on the list field and drag the mouse up or down accordingly.

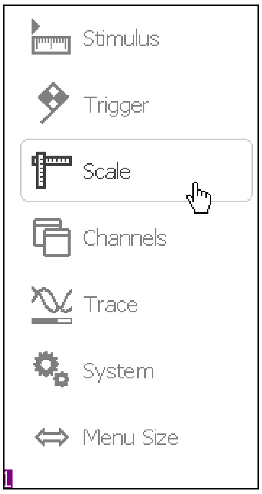
To select the first trace display format click on **Active Trace** and on **Format**. In the **Format** dialog use the following softkeys **SWR > Ok.**

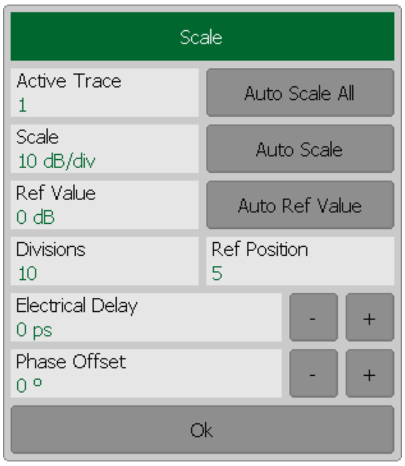
Close the dialogs by **Ok.**

3.6 Trace Scale Setting

For a convenience in operation, change the trace scale using automatic scaling function.

To set the scale of the active trace by the autoscaling function use the following softkeys in the right menu bar **Scale > Auto Scale > Ok.**





The program will automatically set the scale for the best display of the active trace.

If you use the softkeys **Scale > Auto Scale All > Ok**, the program will automatically set the scale for all traces.

Note To activate a trace use the softkey **Active Trace.**

3.7 Analyzer Calibration for Reflection Coefficient Measurement

Calibration of the whole measurement setup, which includes the Analyzer and other devices, supporting connection to the DUT, allows to enhance considerably the accuracy of the measurement.

To perform full 1-port calibration, you need to prepare the kit of calibration standards: OPEN, SHORT and LOAD. Every kit has its description and specifications of the standards.

To perform proper calibration, you need to select the correct kit type in the program. In the process of full 1-port calibration, connect calibration standards to the test port one after another, as shown in Figure 3.2.

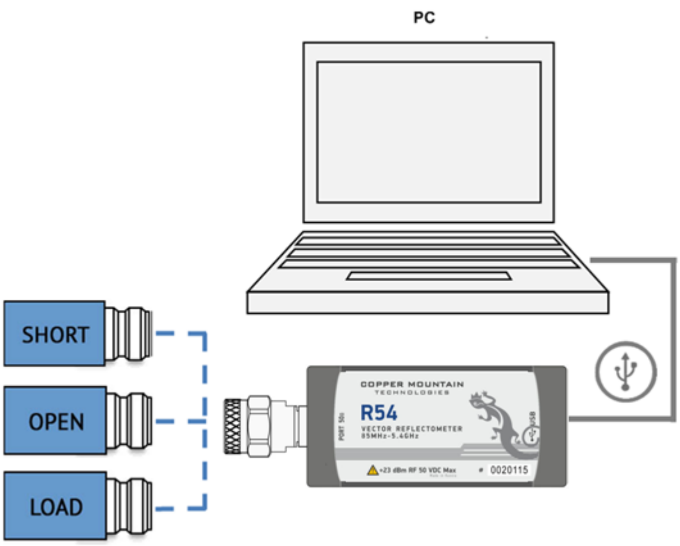
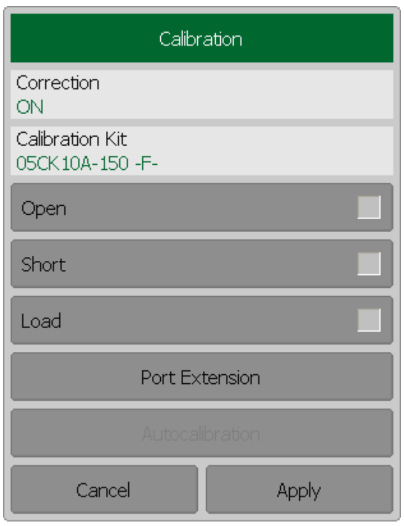
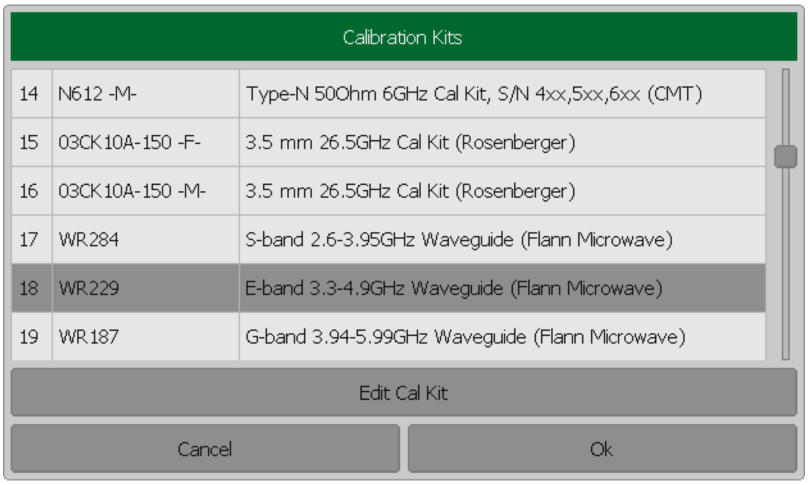


Figure 3.2. Full 1-port calibration circuit

In the current example Agilent 85032B/E calibration kit is used.

To select the calibration kit use the following softkeys in the left menu bar **Calibration > Calibration Kit.**

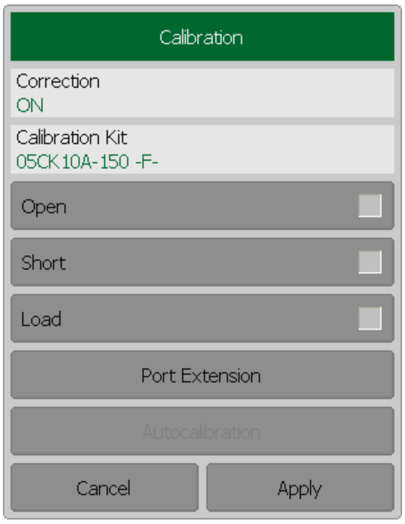


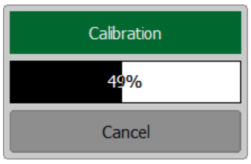


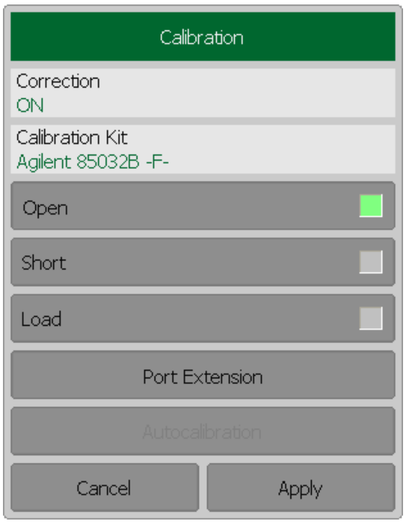
Then select the required kit from the **Calibration Kits** list and complete the setting by **Ok.**

To perform full 1-port calibration you should execute measurements of the three standards. After that the table of calibration coefficients will be calculated and saved into the memory of the Analyzer. Before you start calibration, disconnect the DUT from the Analyzer.

To perform full 1-port calibration use the following softkey in the left menu bar **Calibration.**







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

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

Connect a LOAD standard and click **Load.**

After clicking any of the **Open**, **Short**, or **Load** softkeys, wait until the calibration procedure is completed.

To complete the calibration and calculate the table of calibration coefficients click **Apply.** Then re-connect the DUT to the Analyzer test port.

3.8 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 in Figure 3.3. In the current example, a reflection standard of SWR = 1.2 is used as a DUT.

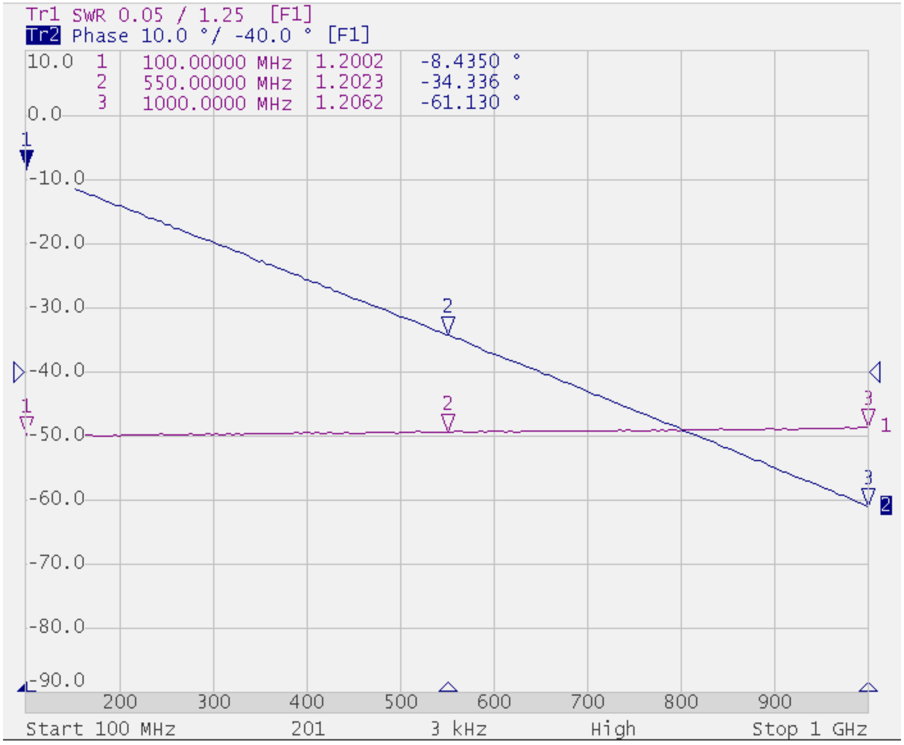
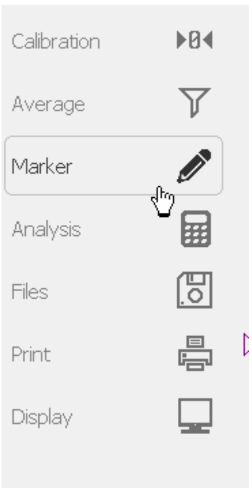
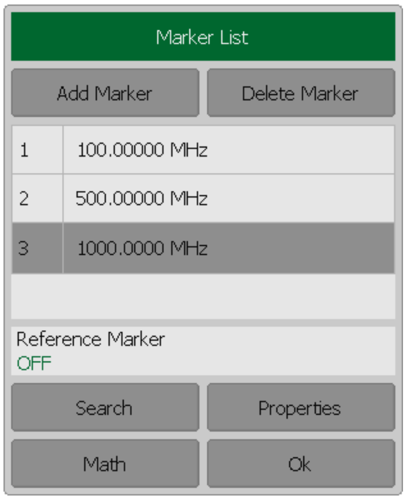


Figure 3.3 SWR and reflection coefficient phase measurement example

---

To enable a new marker use the following softkeys in the left menu bar **Marker > Add Marker.**





Double click on the marker in the **Marker List** to activate the on-screen keypad and enter the marker frequency value.

Complete the setting by **Ok.**

---

4. MEASUREMENT CONDITIONS SETTING

4.1 Screen Layout and Functions

The screen layout is represented in Figure 4.1. In this section you will find detailed description of the softkey menu bars and instrument status bar. The channel windows will be described in the next section.

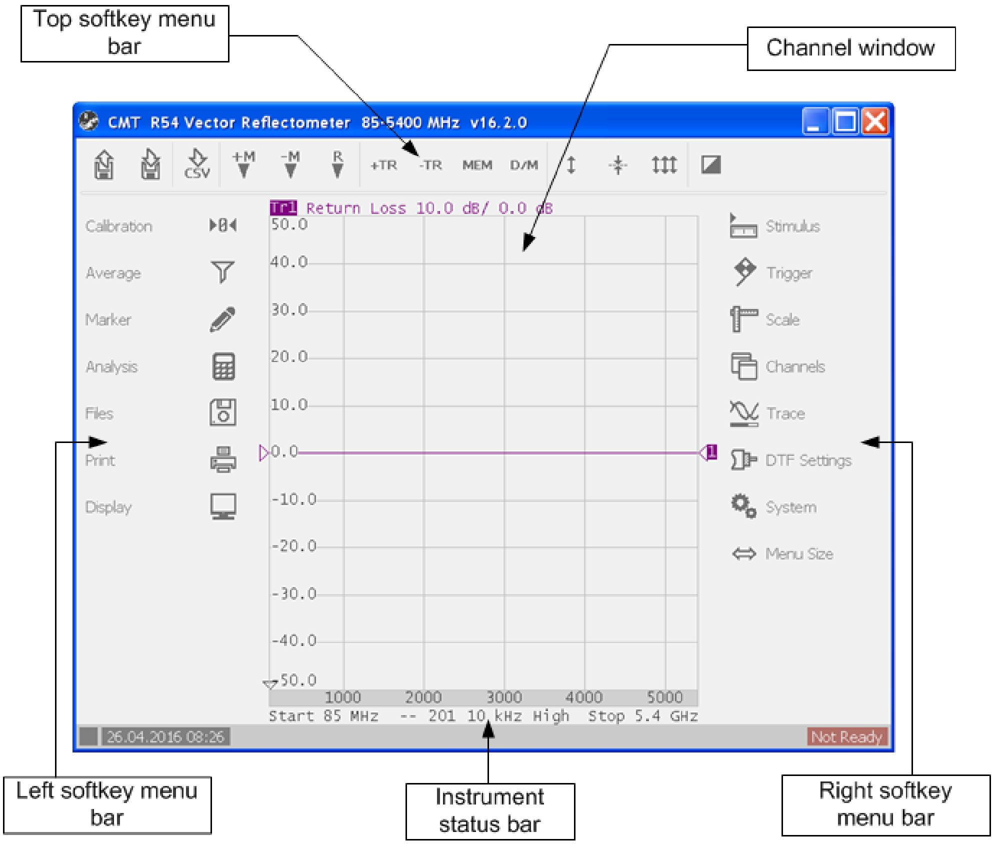


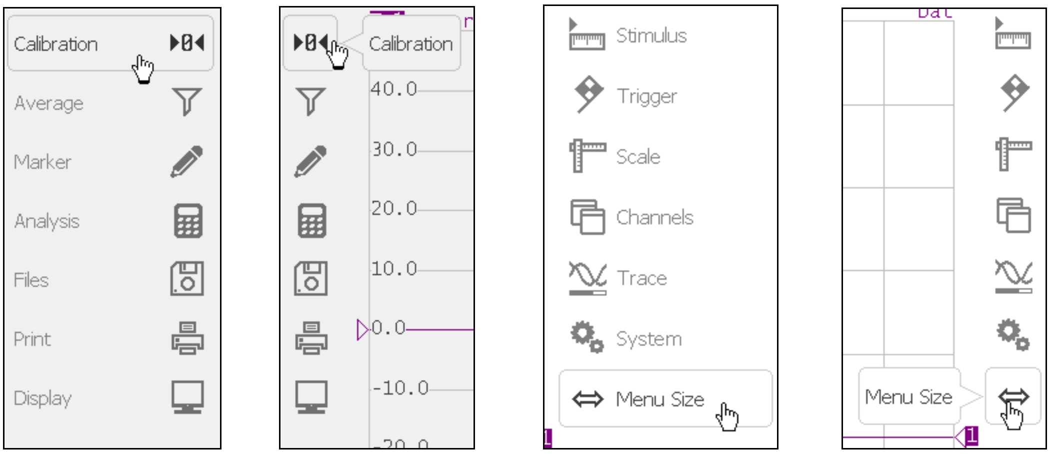
Figure 4.1 Analyzer screen layout

4.1.1 Left and Right Softkey Menu Bars

The softkey menu bars in the left and right parts of the screen are the main menu of the program. Each softkey represents one of the submenus. The menu system is multilevel and allows to access to all the functions of the Analyzer.

You can manipulate the menu softkeys by the mouse or using a touch screen.

On-screen alphanumeric keypads also support data entering from external PC keyboard. Besides, you can navigate the menu by «Up Arrow», «Down Arrow»,«Enter», «Esc» keys on the external keyboard.



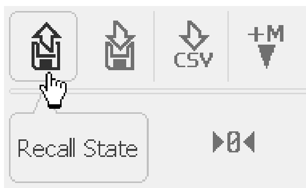
To expand the menu bar click on it and drag the cursor to the right or to the left accordingly. To collapse the menu bar click on it and drag the cursor to the right or to the left accordingly.

You can also click the softkey **Menu Size** to expand or to collaps the menu bar.

4.1.2 Top Menu Bar

The menu bar contains the functions of the most frequently used softkeys.

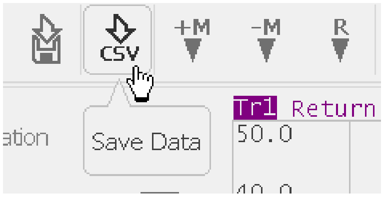
The softkey **Recall State** allows to recall the state from a file of Analyzer state (see section 8.1.2).



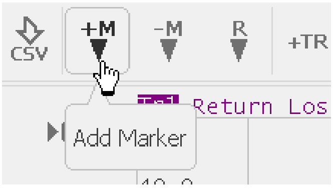
The softkey **Save State** allows to save the Analyzer state (see section 8.1.1).

Note	Type of saving is set by the user in the dialog form **Save type** (see section 8.1)**.**

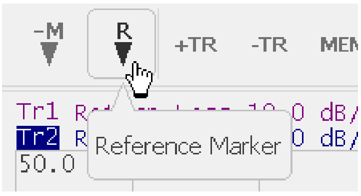
The softkey **Save Data** allows to save the trace data in CSV format (see section 8.3.1).



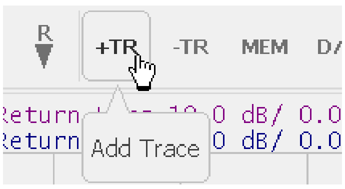
The softkeys **Add Marker** and **Delete Marker** add and delete markers on the trace respectively.

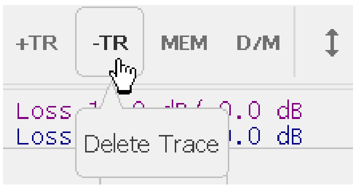


The softkey **Reference Marker** allows to add the reference marker on the trace. To delete the reference marker reclick this key.

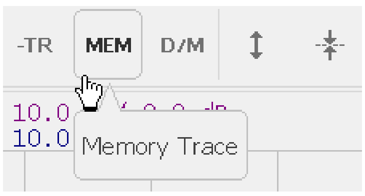


The softkeys **Add Trace** and **Delete Trace** add and delete traces respectively.

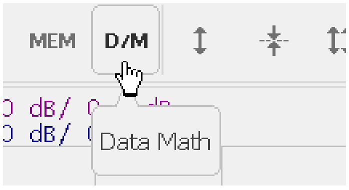


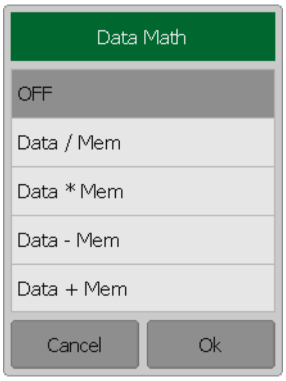


The softkey **Memory trace** enables trace saving into memory (see section 6.2).

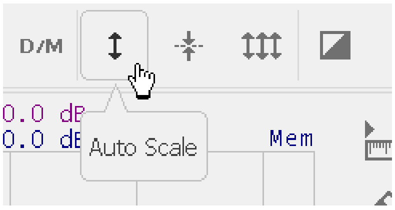


The softkey **Data Math** pops up the corresponding dialog form for choosing the math operation type between data traces and memory traces (see section 6.2.4).

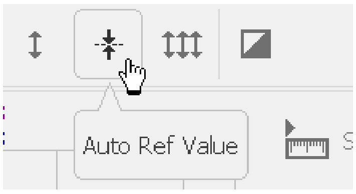




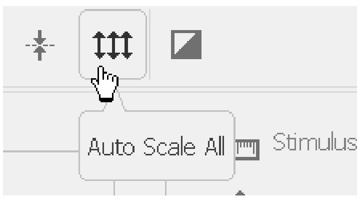
The softkey **Auto Scale** allows to define the trace scale automatically so that the trace of the measured value could fit into the graph entirely (see section 3.6).



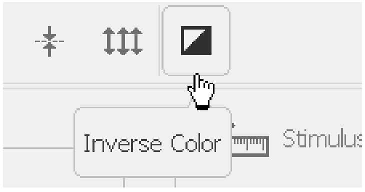
The softkey **Auto Ref Value** executes the automatic selection of the reference level (see section 4.7.6).



The softkey **Auto Scale All** allows to define the trace scale automatically for all traces (see section 3.6).



The softkey **Inverse Color** allows to change the interface color.



4.1.3 Instrument Status Bar

Figure 4.2 Instrument status bar

The instrument status bar is located at the bottom of the screen. It can contain the following messages (see Table 4.1).

Table 4.1 Messages in the instrument status bar

Field Description	Message	Instrument Status
DSP status	**Not Ready**	No communication between DSP and PC.
	Loading	DSP program is loading.
	Ready	DSP is running normally.
	Standby	DSP is in energy saving standby mode.
Sweep status	**Measure**	Continuous sweep.
	Hold	A sweep is on hold.
	External	Waiting for “External” trigger.
	Bus	Waiting for “Bus” trigger.
Factory calibration error	**System Cal Failure**	ROM error of system calibration.

Field Description	Message	Instrument Status
Error correction status	**Correction Off**	Error correction disabled by the user[1].
System correction status	**System Correction Off**	System correction disabled by the user.
Temperature	**20.00˚C**	Internal device temperature. To switch between ˚C/˚F click on the corresponding field.
	68.00˚F	

1 Disabling of error correction does not affect factory calibration.

4.2 Channel Window Layout and Functions

The channel windows display the measurement results in the form of traces and numerical values. The screen can display up to 4 channel windows simultaneously. Each window has the following parameters:

- Frequency range;
- Sweep type;
- Number of points;
- IF bandwidth.

Note	The calibration parameters are applied to the whole Analyzer and affect all the channel windows.

Physical analyzer processes the logical channels in succession.

In turn each channel window can display up to 4 traces of the measured parameters. General view of the channel window is represented in Figure 4.3.

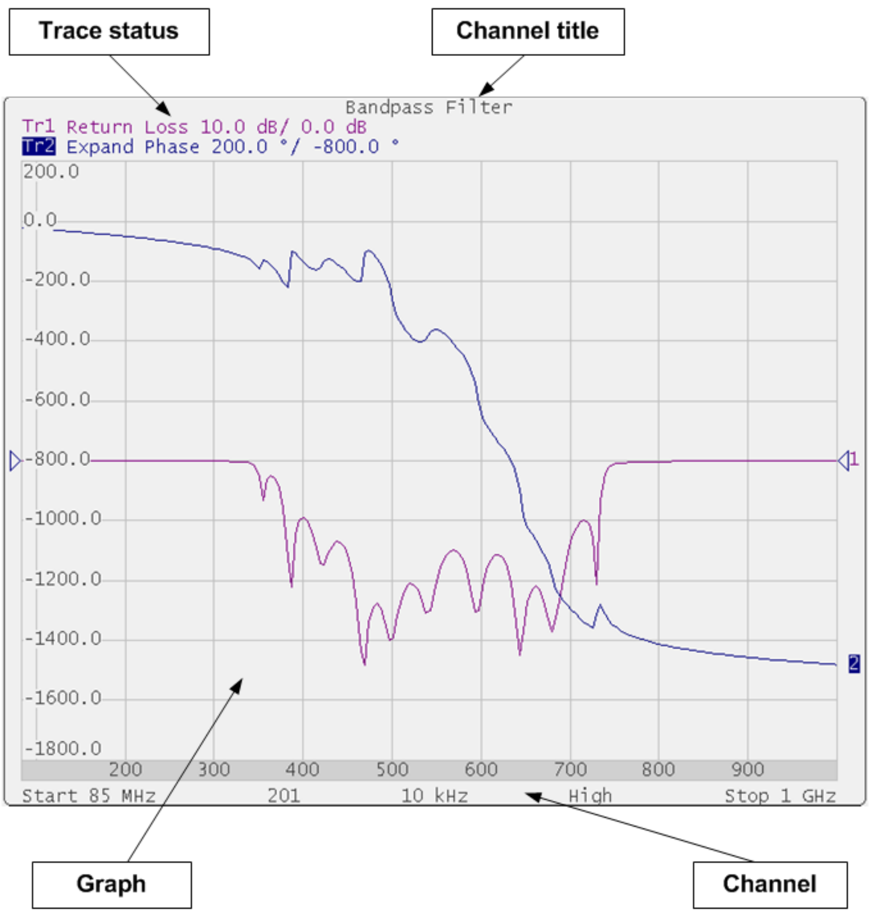
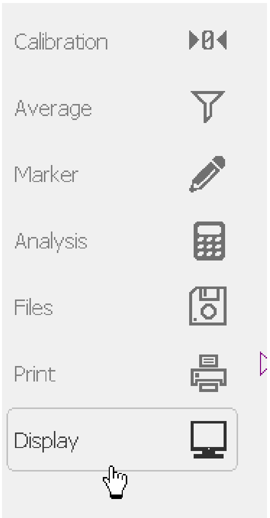


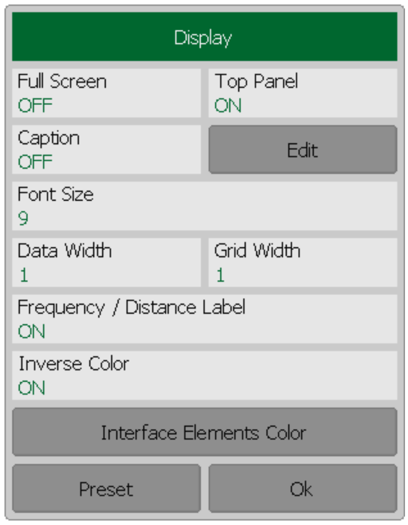
Figure 4.3 Channel window

4.2.1 Channel Title Bar

The channel title feature allows you to enter your comment for each channel window.

To show/hide the channel title bar use the softkey **Display.**





Click on **Caption** field in the opened dialog.

Note	To edit the channel title click on the softkey **Edit** to recall the on-screen keypad.

4.2.2 Trace Status Field

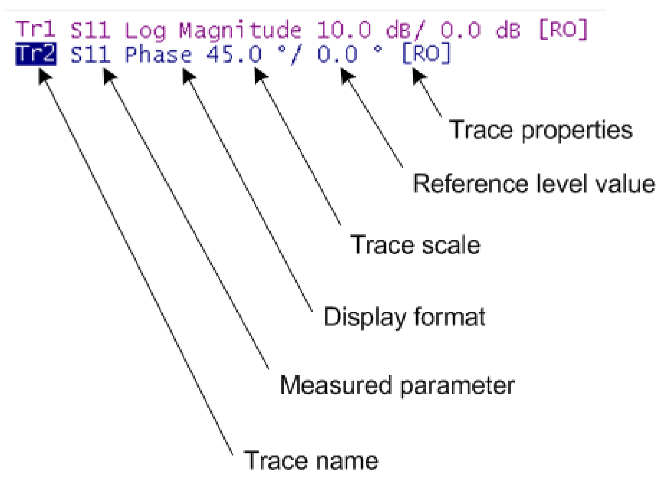


Figure 4.4 Trace status field

The trace status field displays the name and parameters of a trace. The number of lines in the field depends on the number of traces in the channel.

Note	Using the trace status field you can easily modify the trace parameters by the mouse.

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

- Trace name from Tr1 to Tr4. The active trace name is highlighted in inverted color;
- Display format, e.g. Return Loss;
- Trace scale in measurement units per division, e.g. 0.5 dB/;
- Reference level value, e.g. -20.0 dB;
- Trace status is indicated as symbols in square brackets (see Table 4.2).

Table 4.2 Trace status symbols definition

Status	Symbols	Definition
Error Correction	**RO**	OPEN response calibration
	RS	SHORT response calibration
	F1	Full 1-port calibration
Data Analysis	**Z0**	Port impedance conversion
	Dmb	De-embedding
	Emb	Embedding
	Pxt	Port extension
Math Operations	**D+M**	Data + Memory
	D-M	Data - Memory
	D*M	Data * Memory
	D/M	Data / Memory
Maximum Hold	**Max**	Hold of the trace maximum between repeated measurements
Electrical Delay	**Del**	Electrical delay other than zero
Phase Offset	**PhO**	Phase offset value other then zero
Smoothing	**Smo**	Trace smoothing
Gating	**Gat**	Time domain gating
Conversion	**Zr**	Reflection impedance
	Yr	Reflection admittance
	1/S	S-parameter inversion
	Conj	Conjugation

Status	Symbols	Definition
Trace display	**Dat**	Data trace
	Mem	Memory trace
	D&M	Data and memory traces
	Off	Data and memory traces - off

4.2.3 Graph Area

The graph area displays the traces and numeric data (see Figure 4.5).

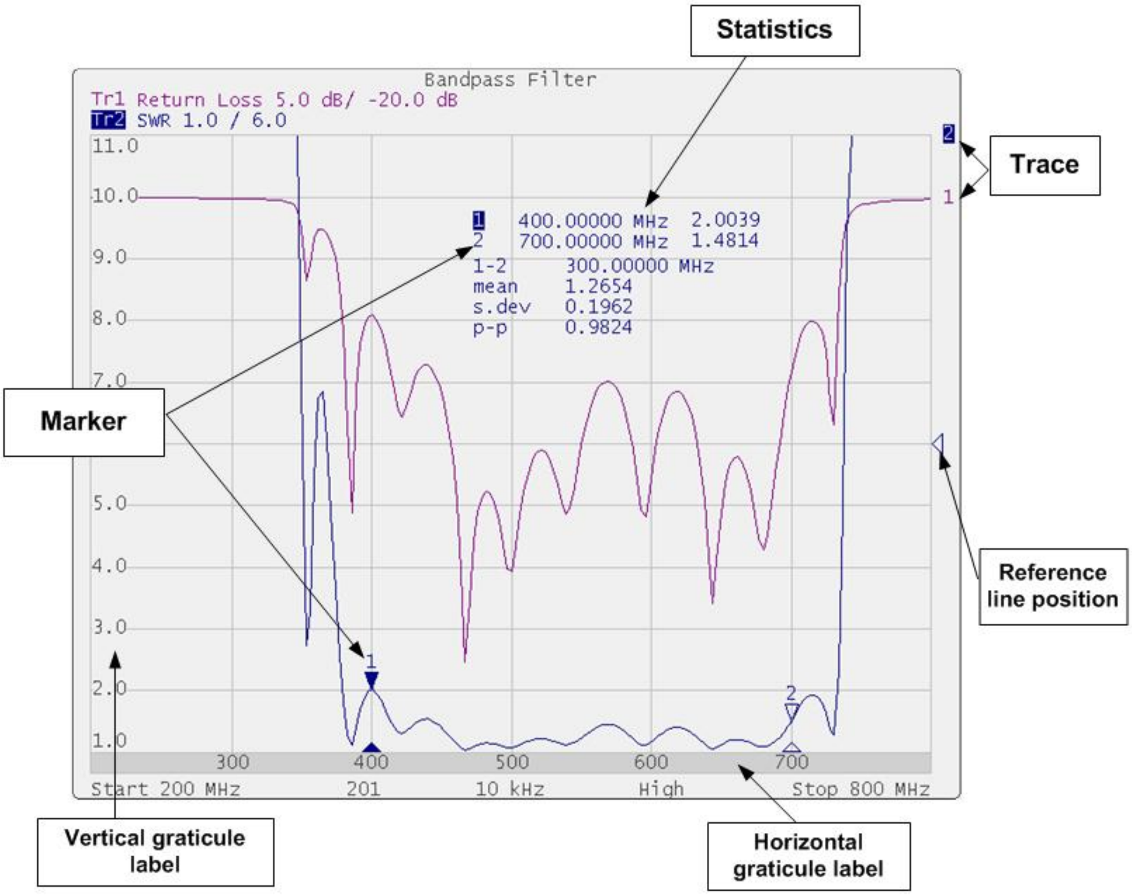


Figure 4.5 Graph area

The graph area contains the following elements:

- **Vertical graticule label** displays the vertical axis numeric data for the active trace;
- **Horizontal graticule label** displays stimulus axis numeric data (frequency, time, or distance);
- **Reference level position** indicates the reference level position of the trace;
- **Markers** indicate the measured values in different points on the active trace. You can enable display of the markers for all the traces simultaneously;
- **Marker functions**: statistics, bandwidth, flatness, RF filter;
- **Trace number** allows trace identification in the channel window;
- **Current stimulus position** indication appears when sweep duration exceeds 1 sec.

Note	Using the graticule labels, you can easily control all the trace parameters by the mouse.

4.2.4 Markers

The markers indicate the stimulus values and the measured values in selected points of the trace (see Figure 4.6).

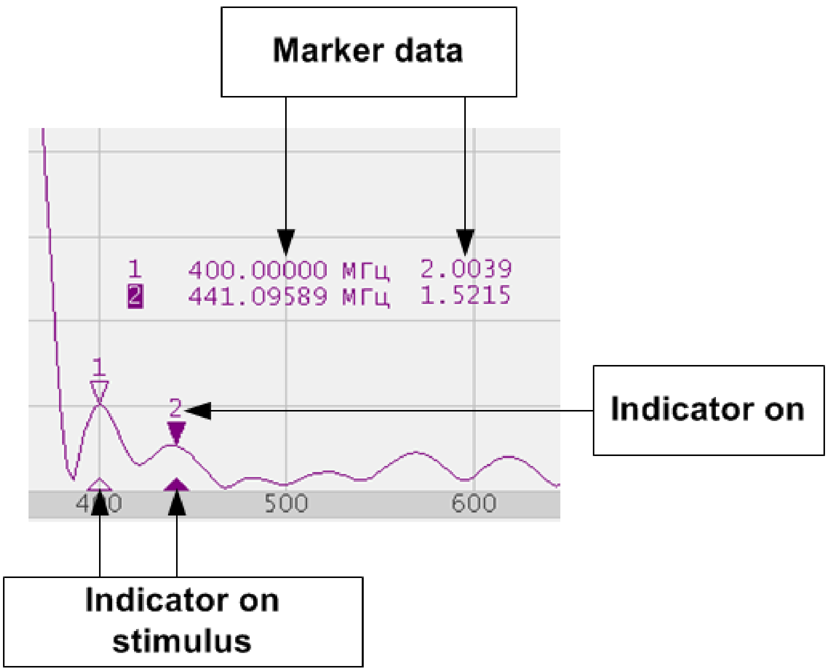


Figure 4.6 Markers

The markers are numbered from 1 to 16. The reference marker is indicated with R symbol. The active marker is indicated in the following manner: its number is highlighted in inverse color, the stimulus indicator is fully colored.

4.2.5 Channel Status Bar

The channel status bar is located in the bottom part of the channel window (see Figure 4.7)

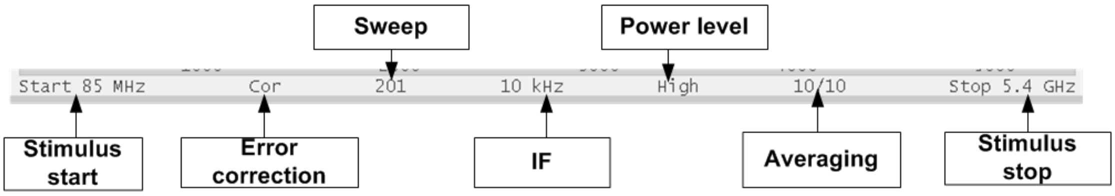


Figure 4.7 Channel status bar

The channel status bar contains the following elements:

- **Stimulus start** field allows to display and enter the start frequency. This field can be switched to indication of stimulus center frequency, in this case the word Start will change to Center;
- **Sweep points** field allows to display and enter the number of sweep points. The number of sweep points can have the following values: 2 - 100001;
- **IF bandwidth** field allows to display and set the IF bandwidth. The values can be set from 10 Hz to 30 kHz (100 kHz);
- **Power level** field allows to display and enter the port output power;
- **Stimulus stop** field allows to display and enter the stop frequency . This field can be switched to indication of stimulus span, in this case the word Stop will change to Span;
- **Error correction** field displays the integrated status of error correction for S-parameter traces. The values of this field are represented in Table 4.3.

Table 4.3 Error correction field

Symbol	Definition
--	No calibration data. No calibration was performed.
Cor	Error correction is enabled. The stimulus settings are the same for the measurement and the calibration.
C?	Error correction is enabled. The stimulus settings are not the same for the measurement and the calibration. Interpolation is applied.

Symbol	Definition
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.

4.3 Quick Channel Setting Using Mouse

This section describes the manipulations, which will enable you to set the channel parameters of R140 fast and easy. When you move a mouse pointer in the channel window field where a channel parameter can be changed, the mouse pointer will change its form and a prompt field will appear.

Note	The manipulations described in this section will help you to perform the most frequently used settings only. All the channel functions can be accessed via the softkey menu.

4.3.1 Active Channel Selection

You can select the active channel window when two or more channel windows are open. The border line of the active window will be highlighted (see Figure 4.8). To activate a channel click in its window.

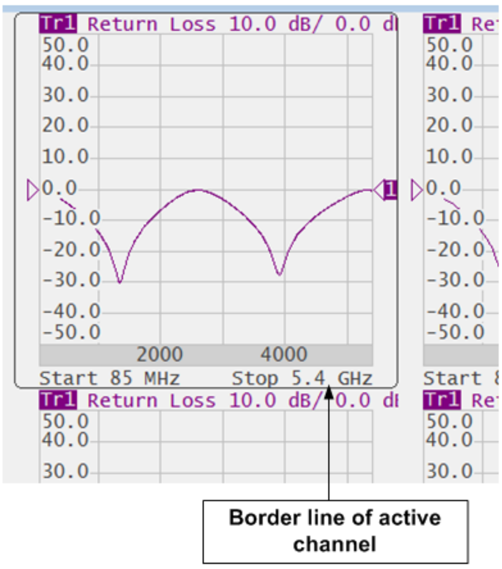
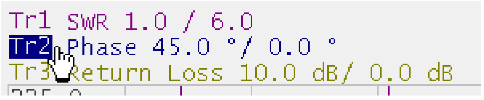


Figure 4.8 Active channel window display

4.3.2 Active Trace Selection

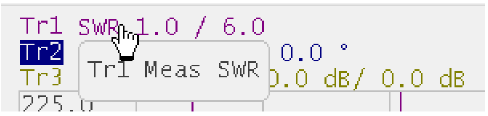
You can select the active trace if the active channel window contains two or more traces.

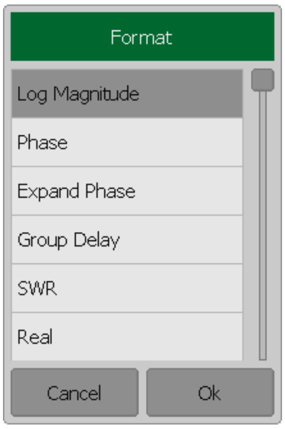


The active trace name will be highlighted in inverted color. In the example given it is Tr2. To activate a trace click on the required trace or its status line.

4.3.3 Display Format Setting

To select the trace display format click on the format name in the trace status line.

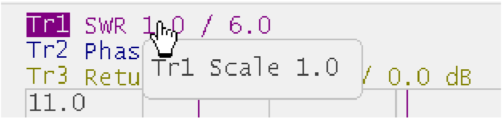


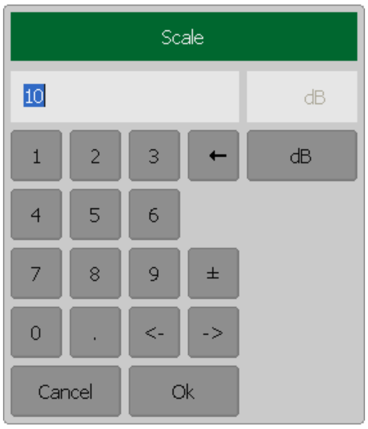


Select the required format in the **Format** dialog and complete the setting by **Ok.**

4.3.4 Trace Scale Setting

To select the trace scale click in the trace scale field of the trace status line.





Enter the required numerical value using the on-screen keypad and complete the setting by **Ok.**

4.3.5 Reference Level Setting

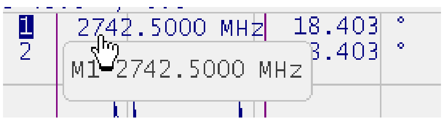
To set the value of the reference level click on the reference level field in the trace status line.

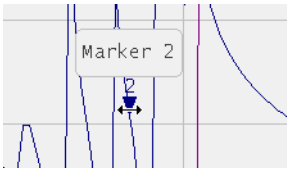
Enter the required numerical value using the on-screen keypad and complete the setting by **Ok.**

4.3.6 Marker Stimulus Value Setting

The marker stimulus value can be set by dragging the marker or by entering the value from the on-screen keypad.

To drag the marker, move the mouse pointer to one of the marker indicators.



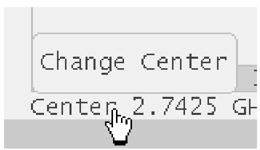


The marker will become active, and a pop-up hint with its name will appear near the marker. The marker can be moved either by dragging its indicator or its hint area.

To enter the numerical value of the stimulus in the marker data click on the stimulus value. Then enter the required value using the on-screen keypad.

4.3.7 Switching between Start/Center and Stop/Span Modes

To switch between the modes Start/Center and Stop/Span click in the respective field of the channel status bar.



Label **Start** will be replaced by **Center**, and label **Stop** will be replaced by **Span**.

4.3.8 Start/Center Value Setting

To enter the **Start/Center** numerical values click on the respective field in the channel status bar.

Then enter the required value using the on-screen keypad.

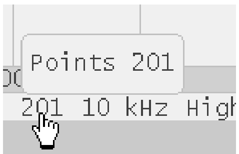
4.3.9 Stop/Span Value Setting

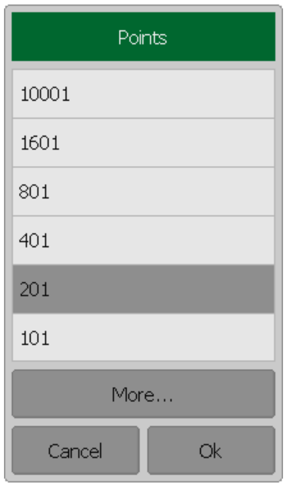
To enter the **Stop/Span** numerical values click on the respective field in the channel status bar.

Then enter the required value using the on-screen keypad.

4.3.10 Sweep Points Number Setting

To enter the number of sweep points click in the respective field of the channel status bar.

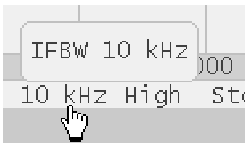


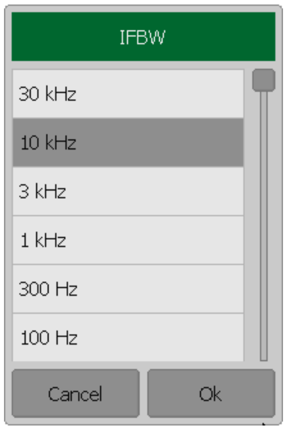


Select the required value in the **Points** dialog and complete the setting by **Ok.**

4.3.11 IF Bandwidth Setting

To set the IF bandwidth click in the respective field of the channel status bar.

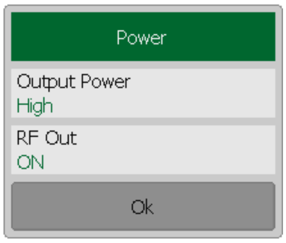




Select the required value in the **IFBW** dialog and complete the setting by **Ok.**

4.3.12 Power Level Setting

To set the output power level click in the respective field of the channel status bar.



This way you can switch between high and low power settings.

4.4 Channel and Trace Display Setting

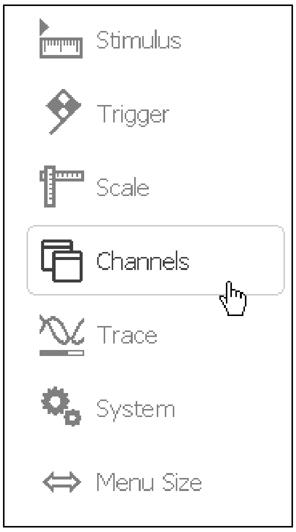
The Analyzer supports 4 channels, which allows measurements with different stimulus parameter settings. The parameters related to a logical channel are listed in Table 4.4

4.4.1 Setting the Number of Channel Windows

A channel is represented on the screen as an individual channel window. The screen can display from 1 to 4 channel windows simultaneously. By default one channel window is opened.

The program supports three options of the channel window layout: one channel, two channels, and four channels. The channels are allocated on the screen according to their numbers from left to right and from top to bottom. If there are more than one channel window on the screen, one of them is selected as active. The border line of the active window will be highlighted in inverted color.

To set the number of channel windows displayed on the screen use the following softkey in the right menu bar **Channels.** Then select the softkey with the required number and layout of the channel windows.



In the **Active Channel** field, you can select the active channel. The repeated clicking will switch the numbers of all channels.

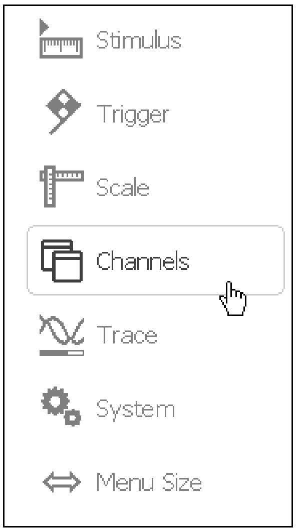
Note	For each open channel window, you should set the stimulus parameters and make other settings. Before you start channel parameter setting or calibration, you need to select this channel as active.

The measurements are executed for open channel windows in succession.

4.4.2 Channel Activating

Before setting channel parameters, you need to activate the channel.

To activate the channel use the following softkeys in the right menu bar **Channels > Active Channel.**



Active Channel field allows viewing the numbers of all channels from 1 to 4. Select the required number of the active channel.

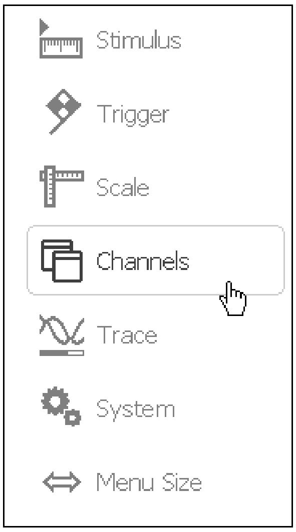
To activate a channel, you can also click on its channel window.

4.4.3 Active Channel Window Maximizing

When there are several channel windows displayed, you can temporarily maximize the active channel window to full screen size.

The other channel windows will be hidden, and this will interrupt the measurements in those channels.

To enable/disable active channel maximizing function use the following softkeys **Channel > Maximize Channel.**



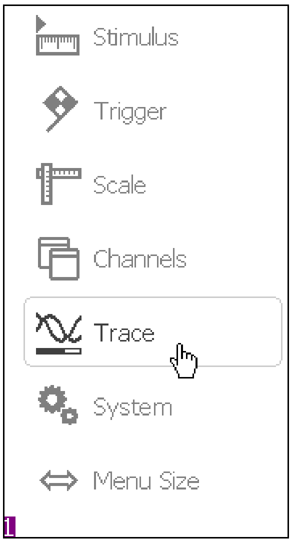
Note	Channel maximizing function can be controlled by a double mouse click on the channel.

4.4.4 Number of Traces Setting

Each channel window can contain up to 4 different traces. Each trace is assigned the display format, scale and other parameters. The parameters related to a trace are listed in Table 4.5.

The traces can be displayed in one graph, overlapping each other, or in separate graphs of a channel window. The trace settings are made in two steps: trace number setting and trace layout setting in the channel window. By default a channel window contains one trace. If you need to enable two or more traces, set the number of traces as described below.

To add a trace use the following softkeys in the right menu bar **Trace > Add Trace.**



To delete a trace use the following softkeys in the right menu bar **Trace > Delete Trace.**

All the traces are assigned their individual names, which cannot be changed. The trace name contains its number. The trace names are as follows: Tr1, Tr2 … Tr4.

Each trace is assigned some initial settings: measured parameter, format, scale and color, which can be modified by the user.

By default the display format for all the traces is set to Return loss (dB).

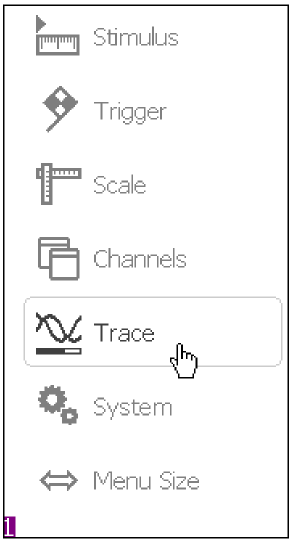
By default the scale is set to 10 dB, reference level value is set to 0 dB, reference level position is in the middle of the graph.

The trace color is determined by its number.

4.4.5 Active Trace Selection

Trace parameters can be entered for the active trace. Active trace belongs to the active channel, and its name is highlighted in inverted color. You have to select the active trace before setting the trace parameters.

To select the active trace use the softkeys in the right menu bar **Trace.**



Click the **Active Trace** to select the trace you want to assign the active.

Note	A trace can be activated by clicking on the trace status bar in the graphical area of the program

Table 4.4 Channel parameters

N	Parameter Description
1	Sweep Range
2	Number of Sweep Points
3	IF Bandwidth

Table 4.5 Trace parameters

N	Parameter Description
1	Display Format
2	Reference Level Scale, Value and Position
3	Electrical Delay, Phase Offset
4	Memory Trace
5	Markers
6	Parameter Transformation

4.5 Measurement Parameters Setting

4.5.1 S-Parameters

For high-frequency network analysis the following terms are applied: incident, reflected and transmitted waves, transferred in the circuits of the setup (see Figure 4.9).

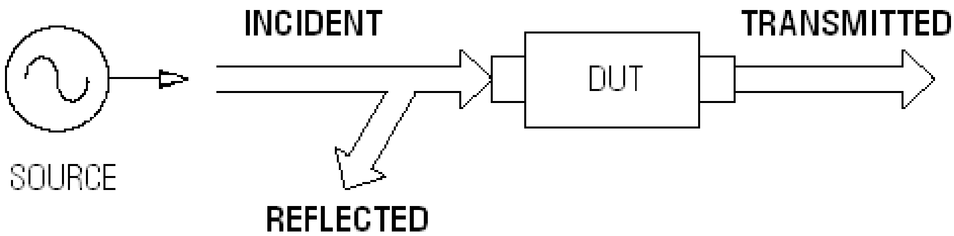


Figure 4.9

Measurement of magnitude and phase of incident, reflected and transmitted signals allow to determin the S-parameters (scattered parameters) of the DUT. An S-parameter is a relation between the complex magnitudes of the two waves:

$$S_{mn} = \frac{transmitted\ wave\ at\ Port\ m}{incident\ wave\ at\ Port\ n}$$

R140 Analyzer has one measurement port which operates as a signal source and as a reflected signal receiver. That is why the Analyzer allows measuring only S11 parameter.

4.5.2 Trace Format

The Analyzer offers the display of the measured S-parameters on the screen in three formats:

- rectangular format;
- polar format;
- Smith chart format.

4.5.3 Rectangular Format

In this format, stimulus values are plotted along X-axis and the measured data are plotted along Y-axis (see Figure 4.10).

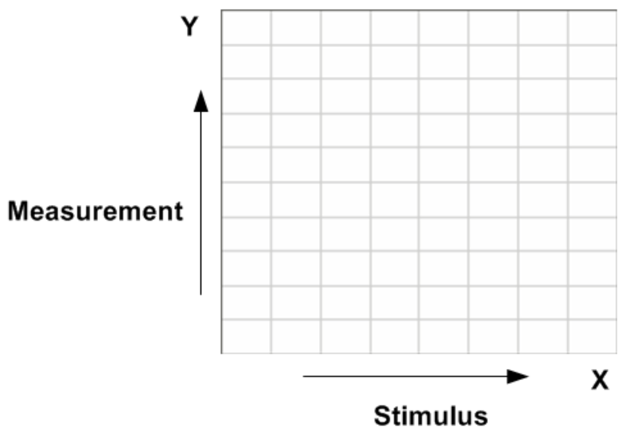


Figure 4.10 Rectangular format

To display S-parameter complex value along Y-axis, it should 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 nine types of rectangular formats depending on the measured value plotted along Y-axis (see Table 4.6).

Table 4.6 Rectangular formats

Format Type Description	Label	Data Type (Y-axis)	Measurement Unit (Y-axis)
Logarithmic Magnitude	**Log Mag**	S-parameter logarithmic magnitude: $20 \cdot \log\lvert S\rvert$, $\lvert S\rvert = \sqrt{a^2 + b^2}$	Decibel (dB)

Format Type Description	Label	Data Type (Y-axis)	Measurement Unit (Y-axis)
Voltage Standing Wave Ratio	**SWR**	$\frac{1+\lvert S\rvert}{1-\lvert S\rvert}$	Abstract number
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: $\sqrt{a^2+b^2}$	Abstract number
Real Part	**Real**	S-parameter real part: $a = re(S)$	Abstract number
Imaginary Part	**Imag**	S-parameter imaginary part: $b = im(S)$	Abstract number
Cable Loss	**Cable Loss**	$A = \frac{1}{2}\cdot(ReturnLoss)$ $A = 10\cdot\log\lvert S\rvert$	Decibel (dB)

4.5.4 Polar Format

Polar format represents the measurement results on the pie chart (see Figure 4.11). The distance to a measured point from the graph center corresponds to the magnitude of its value. The counterclockwise angle from the positive horizontal axis corresponds to the phase of the measured value.

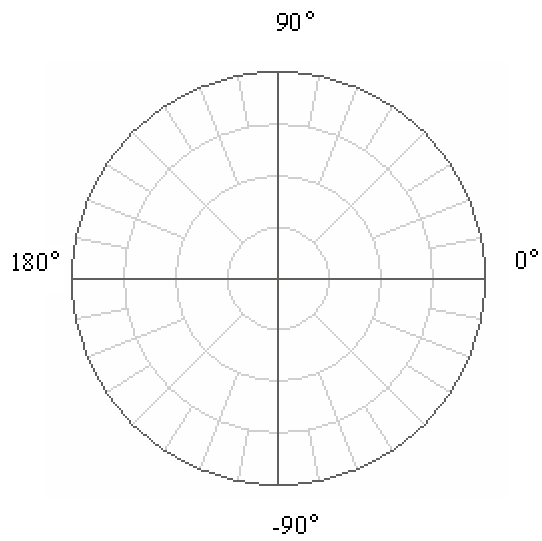


Figure 4.11 Polar format

The polar graph does not have a frequency axis, so frequency will be indicated by the markers. There are three types of polar formats depending on the data displayed by the marker. The traces will remain the same on all the graphs.

Table 4.7 Polar formats

Format Type Description	Label	Data Displayed by Marker	Measurement Unit (Y-axis)
Linear Magnitude and Phase	**Polar (Lin)**	S-parameter linear magnitude	Abstrac number
		S-parameter phase	Degree
Logarithmic Magnitude and Phase	**Polar (Log)**	S-parameter logarithmic magnitude	Decibel (dB)
		S-parameter phase	Degree
Real and Imaginary Parts	**Polar (Re/Im)**	S-parameter real part	Abstract number
		S-parameter imaginary part	Abstract number

4.5.5 Smith Chart Format

Smith chart format is used for representation of impedance values for DUT reflection measurements. In this format, the trace has the same points as in polar format.

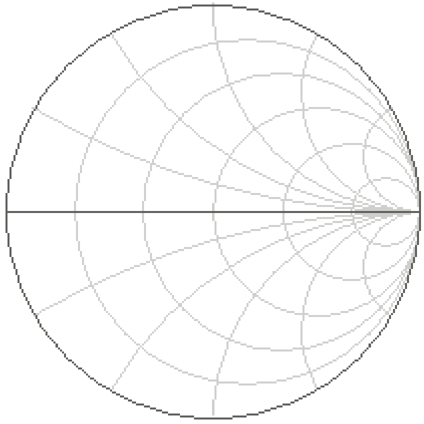


Figure 4.12 Smith chart format

Smith chart format does not have a frequency axis, so frequency will be indicated by the markers. There are five types of Smith chart formats depending on the data displayed by the marker. The traces will remain the same on all the graphs.

Table 4.8 Smith chart format

Format Type Description	Label	Data Displayed by Marker	Measurement Unit (Y-axis)
Linear Magnitude and Phase	**Smith (Lin)**	S-parameter linear magnitude	Abstract number
		S-parameter phase	Degree
Logarithmic Magnitude and Phase	**Smith (Log)**	S-parameter logarithmic magnitude	Decibel (dB)
		S-parameter phase	Degree
Real and Imaginary Parts	**Smith (Re/Im)**	S-parameter real part	Abstract number
		S-parameter imaginary part	Abstract number

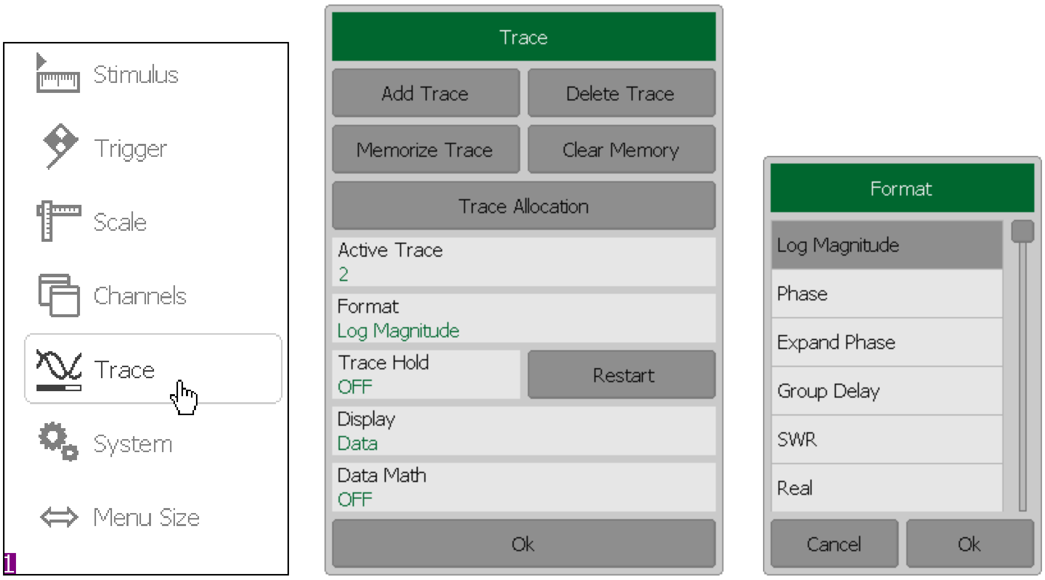
Format Type Description	Label	Data Displayed by Marker	Measurement Unit (Y-axis)
Complex Impedance (at Input)	**Smith (R + jX)**	Resistance at input: $R = re(Z_{inp})$, $Z_{inp} = Z_0 \frac{1+S}{1-S}$	Ohm (Ω)
		Reactance at input: $X = im(Z_{inp})$	Ohm (Ω)
		Equivalent capacitance or inductance: $C = -\frac{1}{\omega X}, \quad X < 0$ $L = \frac{X}{\omega}, \quad X > 0$	Farad (F) Henry (H)
Complex admittance (at Input)	**Smith (G + jB)**	Conductance at input: $G = re(Y_{inp})$, $Y_{inp} = \frac{1}{Z_0} \cdot \frac{1-S}{1+S}$	Siemens (S)
		Susceptance at input: $B = im(Y_{inp})$	Siemens (S)
		Equivalent capacitance or inductance: $C = \frac{B}{\omega}, \quad B > 0$ $L = -\frac{1}{\omega B}, \quad B < 0$	Farad (F) Henry (H)

Z_0 – test port impedance. Z_0 setting is described in section 5.2.6

4.5.6 Data Format Setting

You can select the format for each trace of the channel individually. Before you set the format, first activate the trace.

To set the trace display format use the following softkey in the right menu bar **Trace.**



In the **Trace** dialog select the required trace from **Active Trace** and click on **Format**.

Then select the required format in the **Format** dialog. Complete the setting by **Ok.**

4.6 Trigger Setting

The **trigger mode** determines the sweep actuation of the channel at a trigger signal detection. A channel can operate in one of the following three trigger modes:

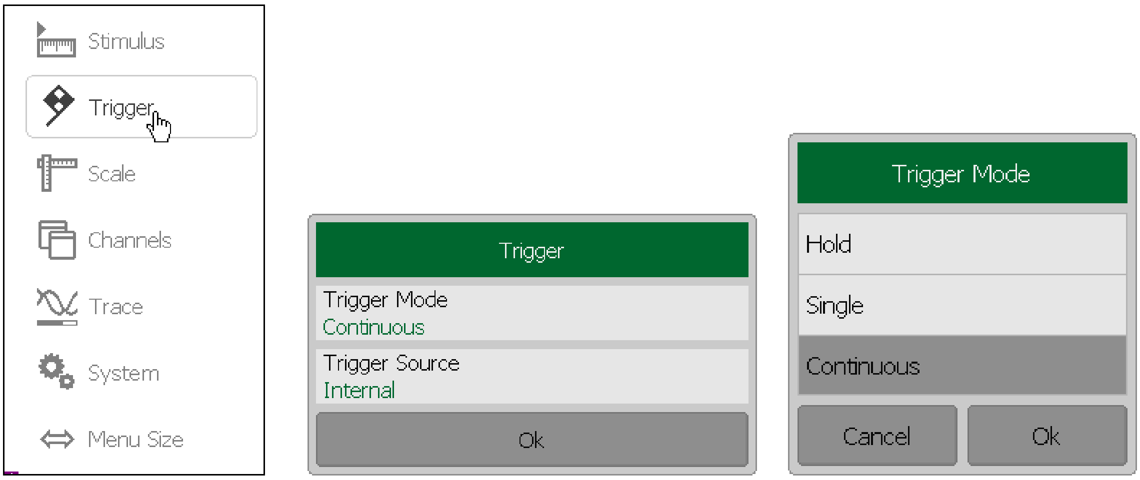
- **Continuous** – a sweep actuation occurs every time a trigger signal is detected;
- **Single** – one sweep actuation occurs with trigger signal detection after the mode has been enabled; after the sweep is complete the channel modes changes to hold;
- **Hold** – sweep actuation is off in the channel, trigger signals do not affect the channel.

The trigger signal applies to the whole Analyzer and controls the trigging of all the channels in the following manner. If more than one channel window are open, the trigger activates successive measurements of all the channels which are not in hold mode. Before measurement of all channels is complete, all additional triggers are ignored. When measurement of all the channels is complete, if there is as least one channel in continuous trigger mode, the Analyzer will enter waiting for a trigger state.

The **trigger source** can be selected by the user from the following four available options:

- **Internal** – the next trigger signal is generated by the Analyzer on completion of each sweep;
- **External** – the external trigger input is used as a trigger signal source (except R54);
- **Bus** – the trigger signal is generated by a command communicated from an external computer from a program controlling the Analyzer via COM/DCOM.

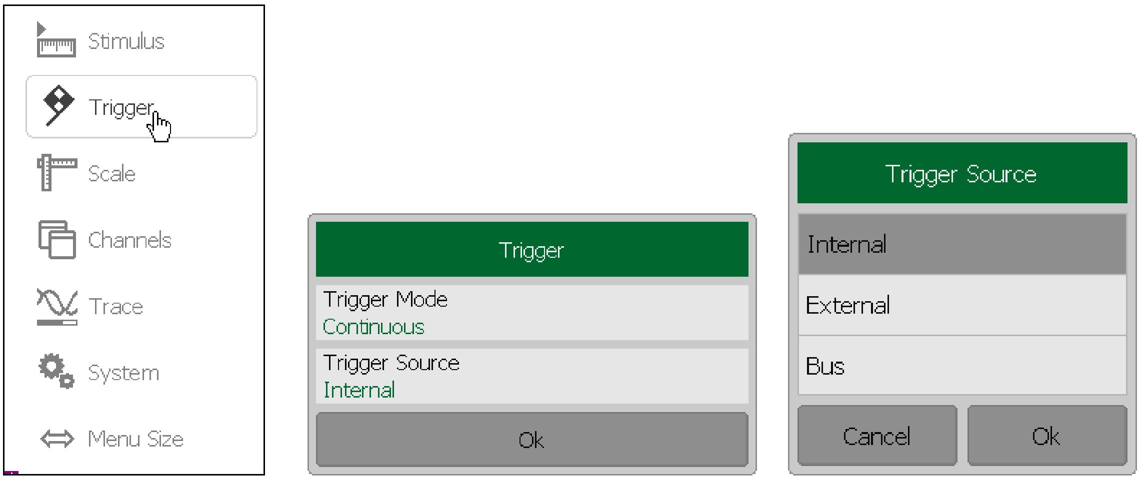
To set the trigger mode, use the following softkeys **Trigger > Trigger Mode**.



Then select the required trigger mode:

- Hold
- Single
- Continuous

To set the trigger source, use the following softkeys **Trigger > Trigger Source**.



Then select the required trigger source:

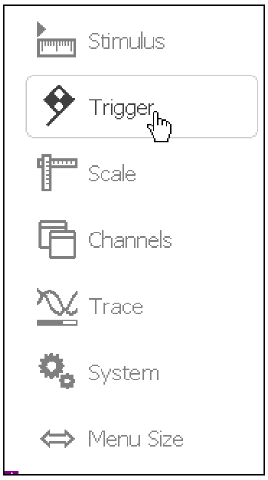
- Internal
- External
- Bus

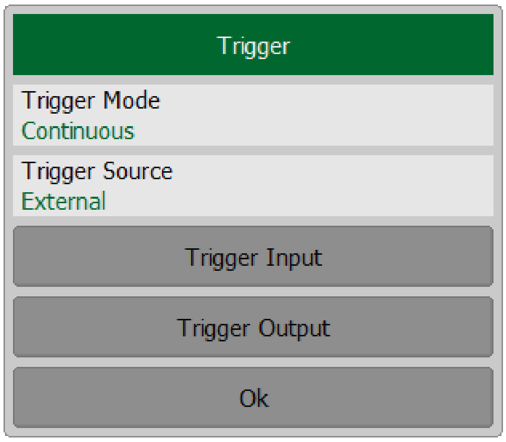
4.6.1 External Trigger (except R54)

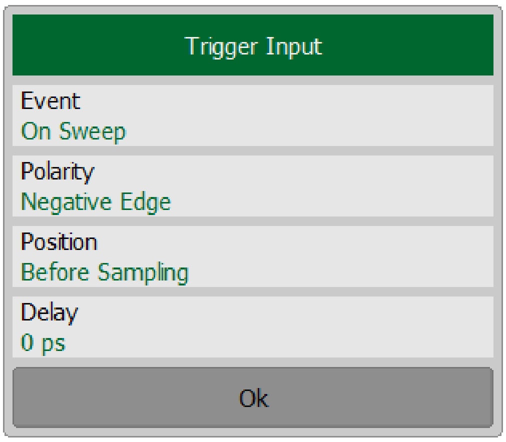
4.6.1.1 Point Feature

By default the external trigger initiates a sweep measurement upon every trigger event (See Figure 4.13 a, b). For the external trigger source, the point trigger feature instead initiates a point measurement upon each trigger event (See Figure 4.13 c, d).

To enable the point trigger feature for external trigger source, use the following softkeys **Trigger > Trigger Input > Event { On Sweep | On Point }.**

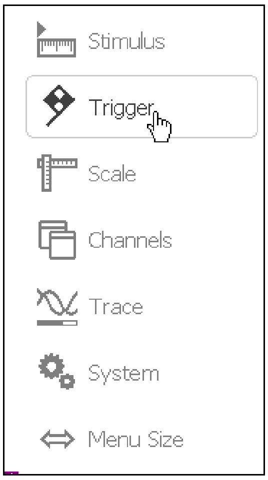


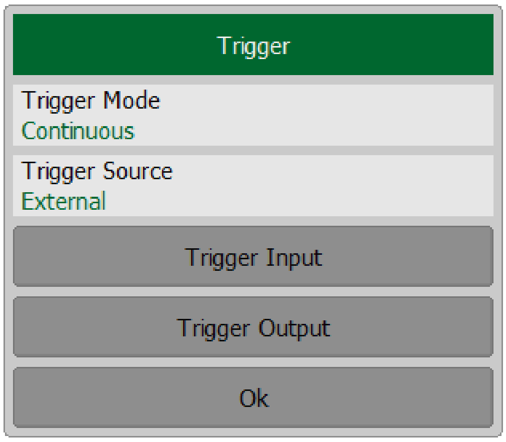


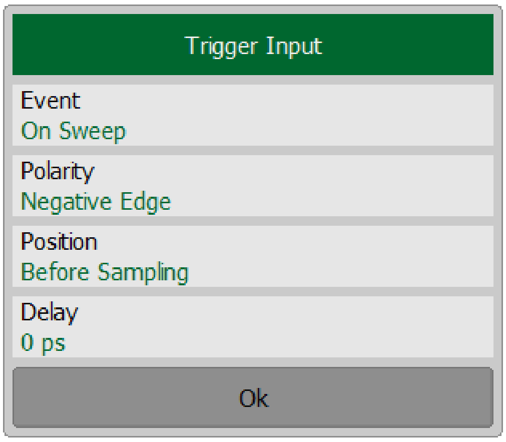


4.6.1.2 External Trigger Polarity

To select the external trigger polarity, use the following softkeys **Trigger > Trigger Input > Polarity { NegativeEdge | Positive Edge }.**







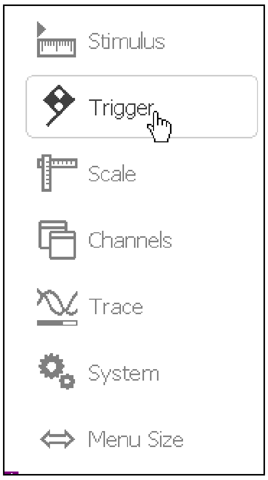
4.6.1.3 External Trigger Position

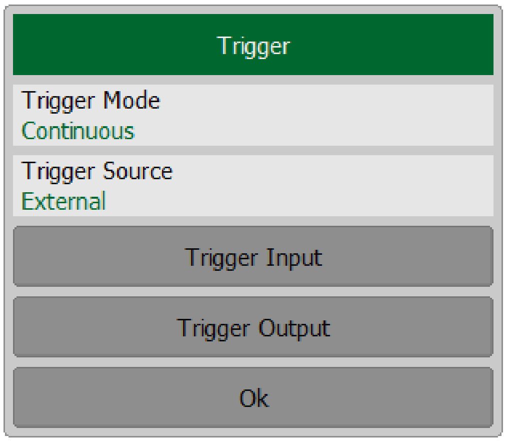
The external trigger position selects the position when Analyzer expects the external trigger signal:

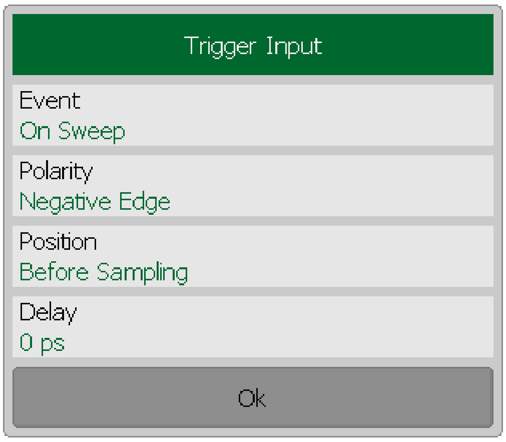
- Before sampling, when the frequency of the stimulus port have been set. The frequency change of the stimulus port begins after sampling (See Figure 4.13 a, c).
- Before the frequency setup and subsequent measurement. The frequency change of the stimulus port begins when the external trigger arrives (See Figure 4.13 b, d).

Depending on the Point Feature settings the external trigger is expected before each point or before the first point of the full sweep cycle.

To select external trigger polarity, use the following softkeys: **Trigger > Trigger Input > Position { Before Sampling | Before Setup }.**



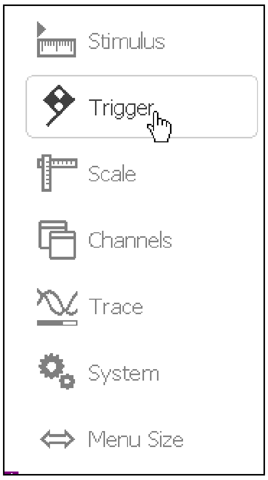


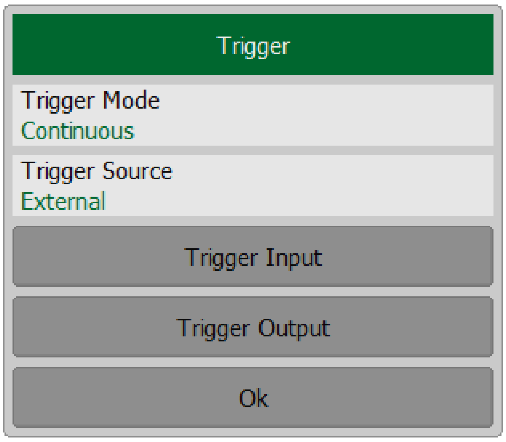


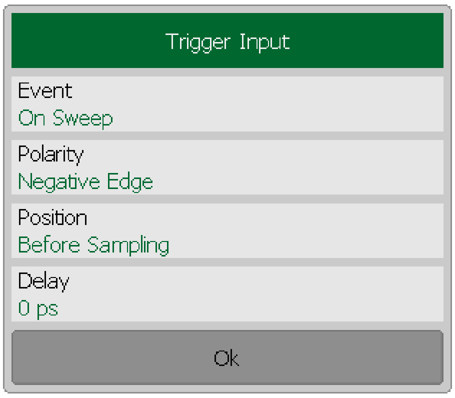
4.6.1.4 External Trigger Delay

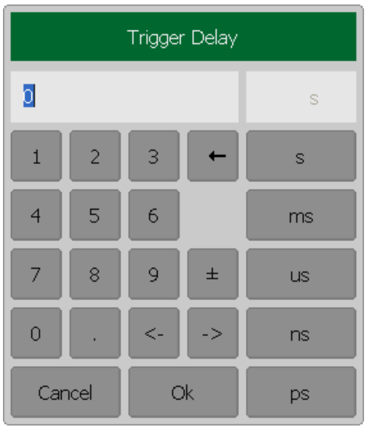
The external trigger delay sets the response delay with respect to the external trigger signal (see Figure 4.13). The delay value has range from 0 to 100 sec with resolution 0.1 μsec.

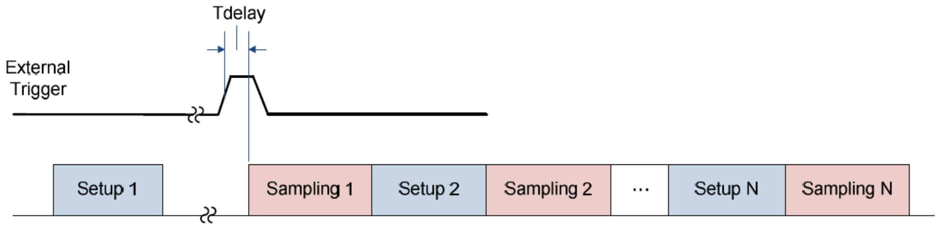
To set the external trigger delay, use the following softkeys: **Trigger > Trigger Input > Delay.**


Stimulus
Trigger
Scale
Channels
Trace
System
Menu Size

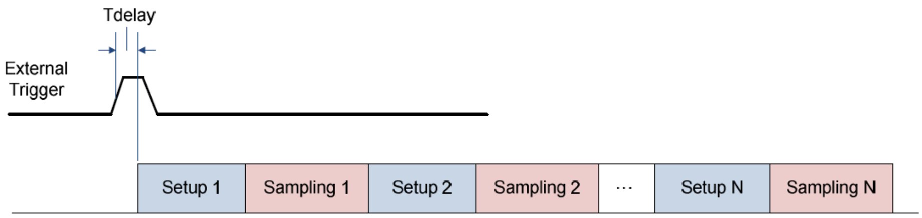

Trigger
Trigger Mode
Continuous
Trigger Source
External
Trigger Input
Trigger Output
Ok


Trigger Input
Event
On Sweep
Polarity
Negative Edge
Position
Before Sampling
Delay
0 ps
Ok

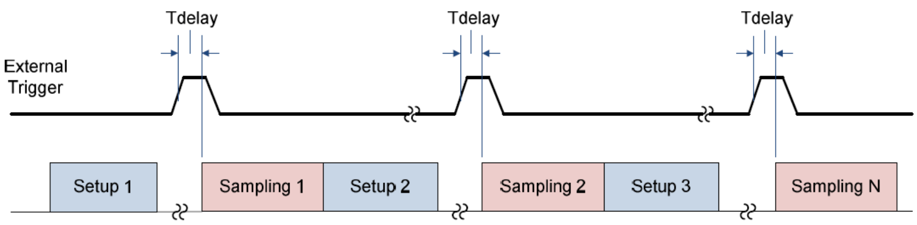

Trigger Delay
0
s
1 2 3 ← s
4 5 6 ms
7 8 9 ± us
0 . <- -> ns
Cancel Ok ps



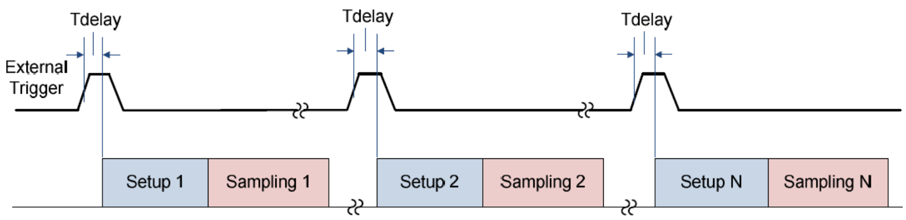
a. Before Sampling, Point trigger OFF



b. Before Setup, Point trigger OFF



c. Before Sampling, Point trigger ON



d. Before Setup, Point trigger ON

Figure 4.13 External Trigger

4.6.2 Trigger Output (except R54/R140)

The trigger output outputs various waveforms depending on the setting of the Output Trigger Function:

- Before frequency setup pulse;
- Before sampling pulse;
- After sampling pulse;
- Ready for external trigger;
- End of sweep pulse;
- Measurement sweep.

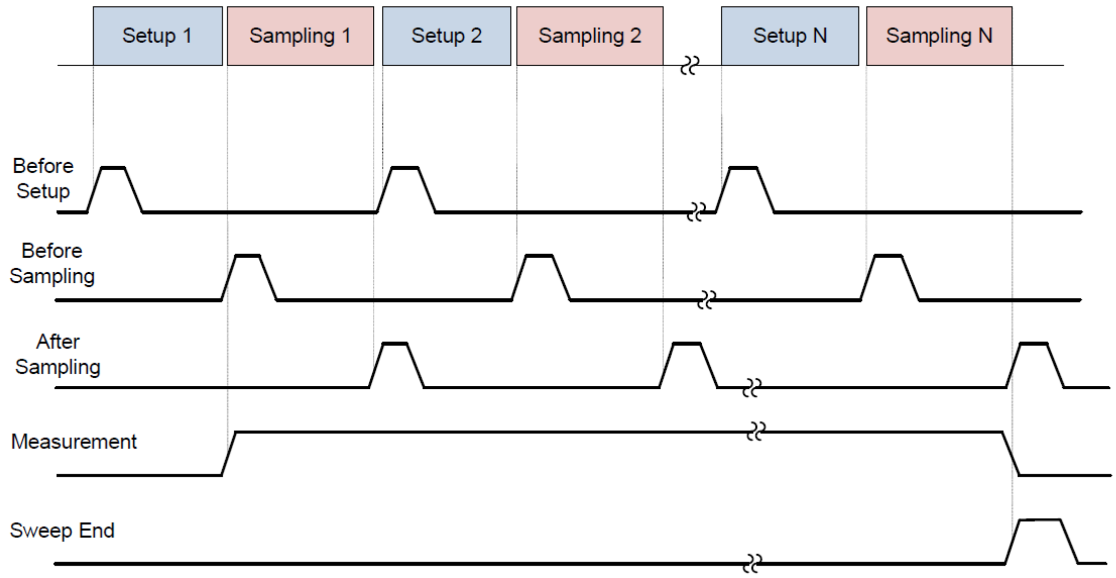
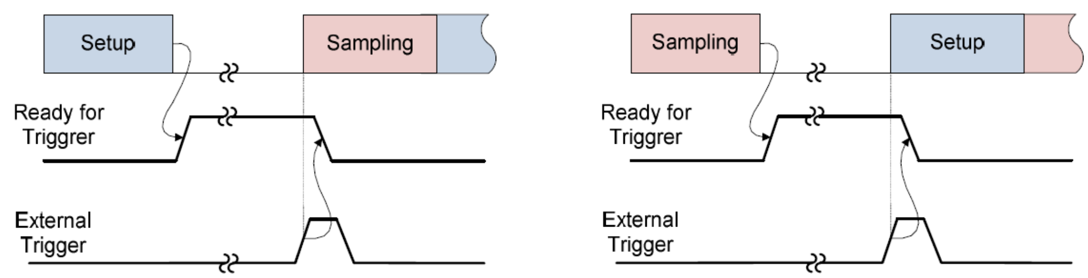


Figure 4.14 Trigger Output (except Ready for Trigger)

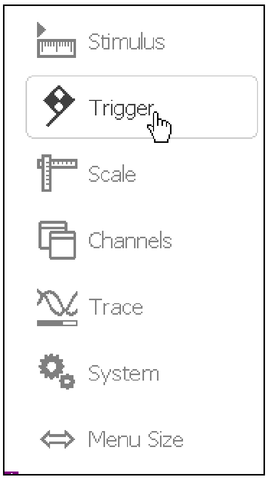


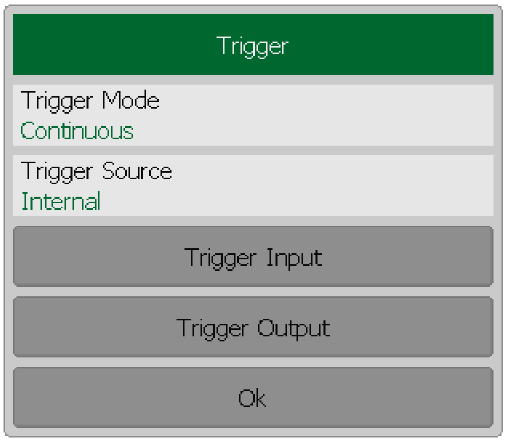
a. External trigger set before sampling   b. External trigger set before setup

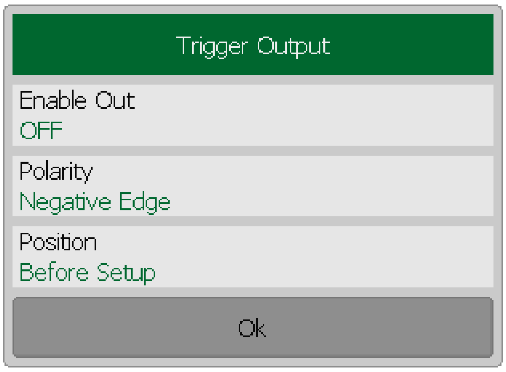
Figure 4.15 Trigger Output (Ready for Trigger)

4.6.2.1 Switching ON/OFF Trigger Output

To enable/disable the trigger output, use the following softkeys **Trigger > Trigger Output > Enable Out.**



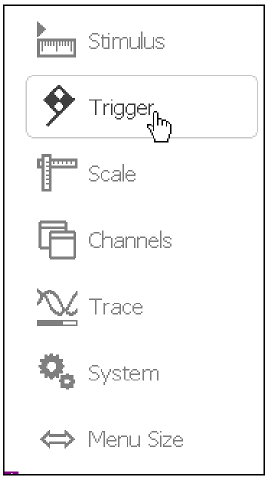


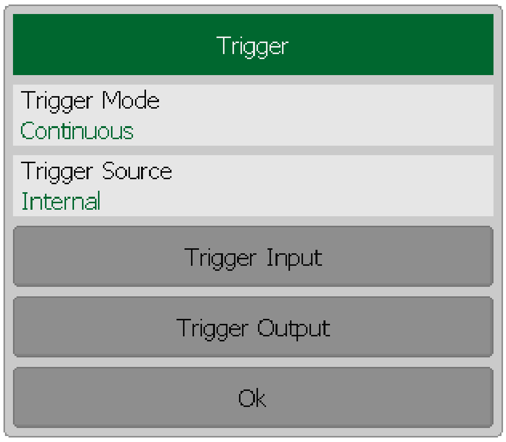


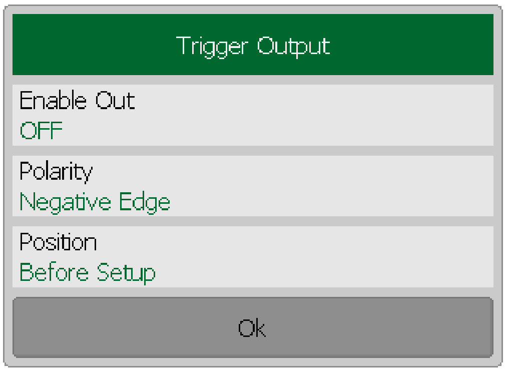
Note	When the Ready for Trigger function of the trigger output is selected the trigger source must be set to external to enable the output trigger.

4.6.2.2 Trigger Output Polarity

To select the polarity of the trigger output, use the following softkeys **Trigger > Trigger Output > Polarity { NegativeEdge | Positive Edge }.**

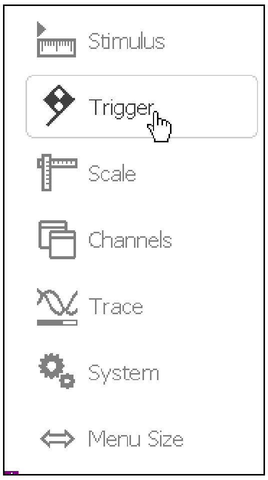


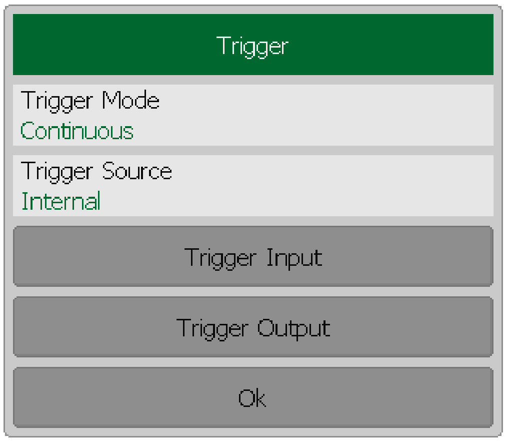


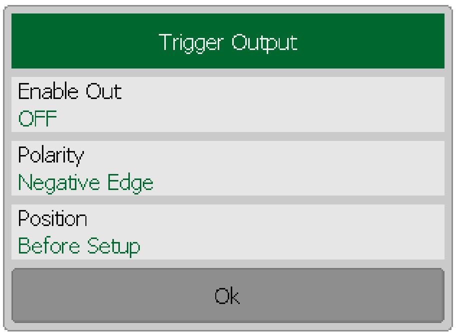


4.6.2.3 Trigger Output Function

To select the function of the trigger output (See Figure 4.14, Figure 4.15), use the following softkeys **Trigger > Trigger Output > Position { Before Setup | BeforeSampling | After Sampling | Ready for Trigger | Sweep End | Measurement }.**







4.7 Scale Setting

4.7.1 Rectangular Scale

For rectangular format you can set the following parameters (see Figure 4.16):

- Trace scale;
- Reference level value;
- Reference level position;
- Number of scale divisions.

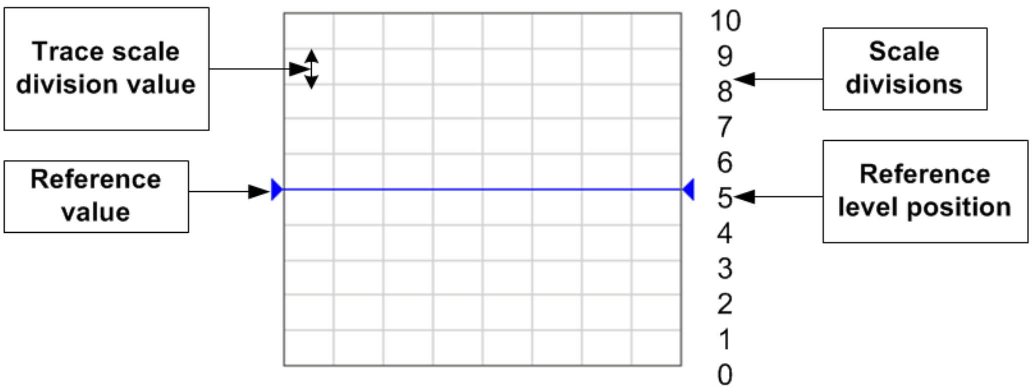
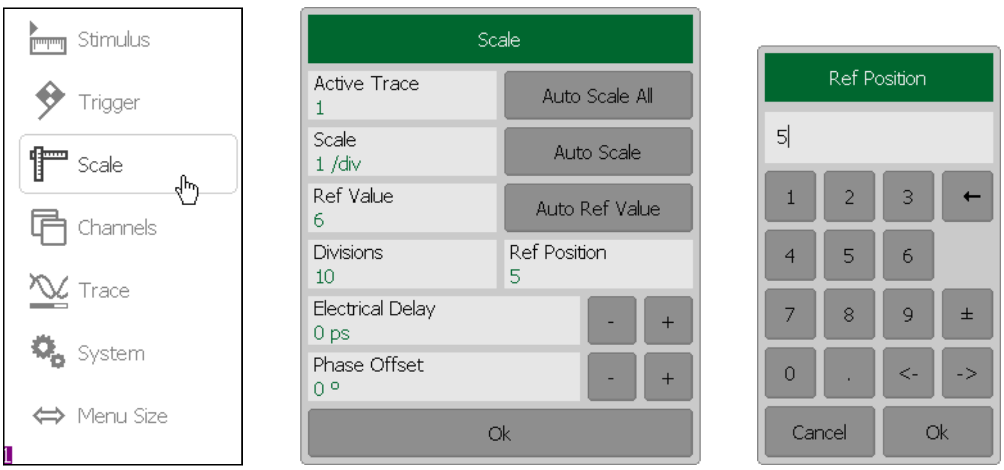


Figure 4.16 Rectangular scale

4.7.2 Rectangular Scale Setting

You can set the scale for each trace of a channel. Before you set the scale, first activate the trace.

To set the scale of a trace use the following softkey in the right menu bar **Scale.**



Then select the **Scale** field and enter the required value using the on-screen keypad.

To set the reference level select the **Ref. Value** field and enter the required value using the on-screen keypad.

To set the position of the reference level select the **Ref. Position** field and enter the required value using the on-screen keypad.

To set the number of trace scale divisions[1] select the **Divisions** field and enter the required value using the on-screen keypad.

4.7.3 Circular Scale

For polar and Smith chart format, you can set the outer circle value (see Figure 4.17).

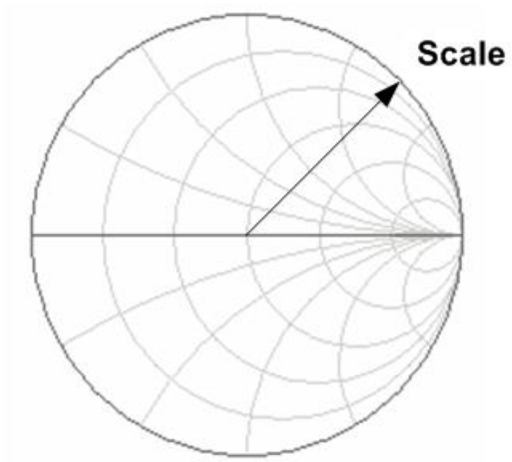
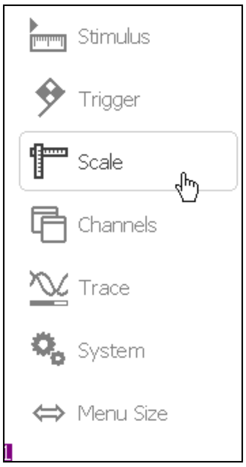


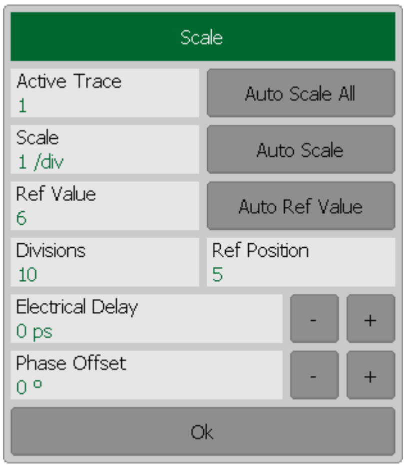
Figure 4.17 Circular scale

4.7.4 Circular Scale Setting

1 The number of scale divisions affects all channel traces.

To set the scale of the circular graph use the following softkey in the right menu bar **Scale.**





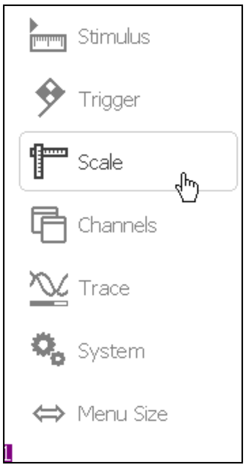
Then select the **Scale** field and enter the required value using the on-screen keypad.

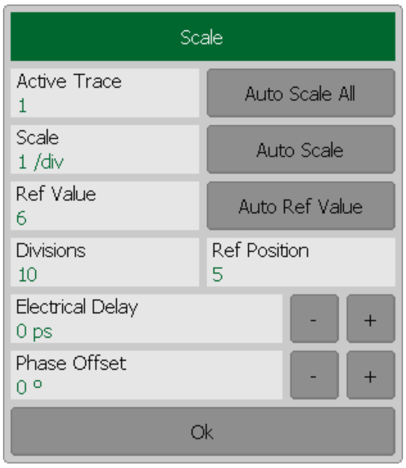
4.7.5 Automatic Scaling

The automatic scaling function allows the user to define the trace scale automatically so that the trace of the measured value could fit into the graph entirely.

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

To execute the automatic scaling use the following softkeys in the right menu bar **Scale > Auto Scale.**



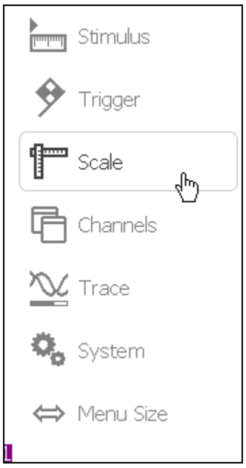


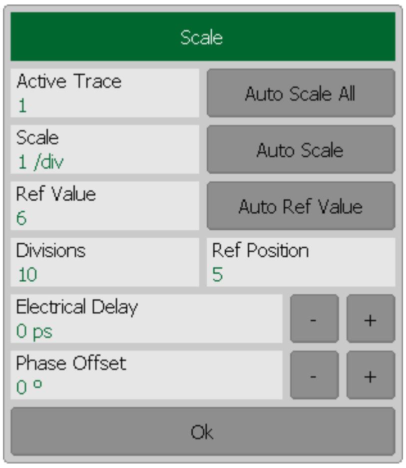
4.7.6 Reference Level Automatic Selection

This function executes automatic selection of the reference level in rectangular coordinates.

After the function has been executed, the trace of the measured value makes the vertical shift so that the reference level crosses the graph in the middle. The scale will remain the same.

To execute the automatic selection of the reference level use the following softkeys in the right menu bar **Scale > Auto Ref. Value.**





4.7.7 Electrical Delay Setting

The electrical delay function allows the user to define the compensation value for the electrical delay of a device. This value is used as compensation for the electrical delay during non-linear phase measurements. The electrical delay is set in seconds.

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

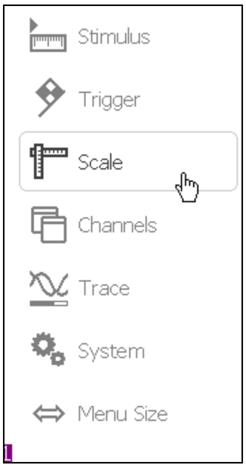
$$S = S \cdot e^{j \cdot 2\pi \cdot f \cdot t}, \quad \text{where}$$

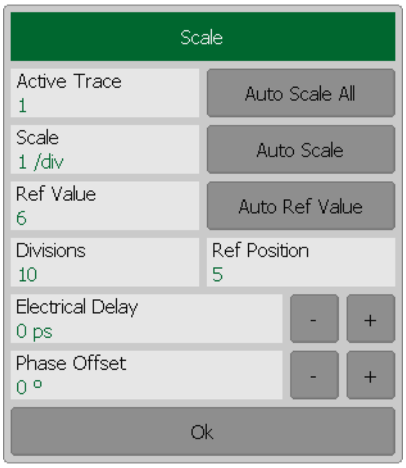
f – frequency, Hz,

t – electrical delay, sec.

The electrical delay is set for each trace individually. Before you set the electrical delay, first activate the trace.

To set the electrical delay use the following softkey in the right menu bar **Scale.**



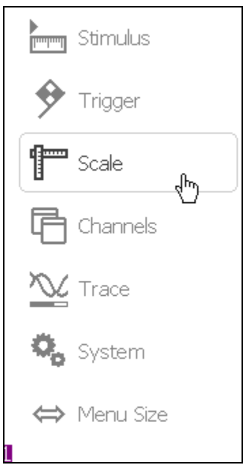


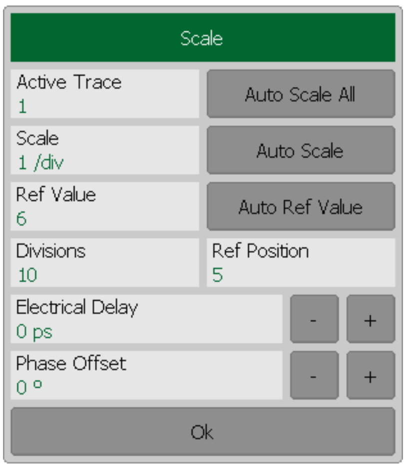
Then select the **Electrical Delay** field and enter the required value using the on-screen keypad.

4.7.8 Phase Offset Setting

The phase offset function allows the user to define the constant phase offset of a trace. The value of the phase offset is set in degrees for each trace individually. To set the phase offset, first activate the trace.

To set the phase offset use the following softkey in the right menu bar **Scale.**





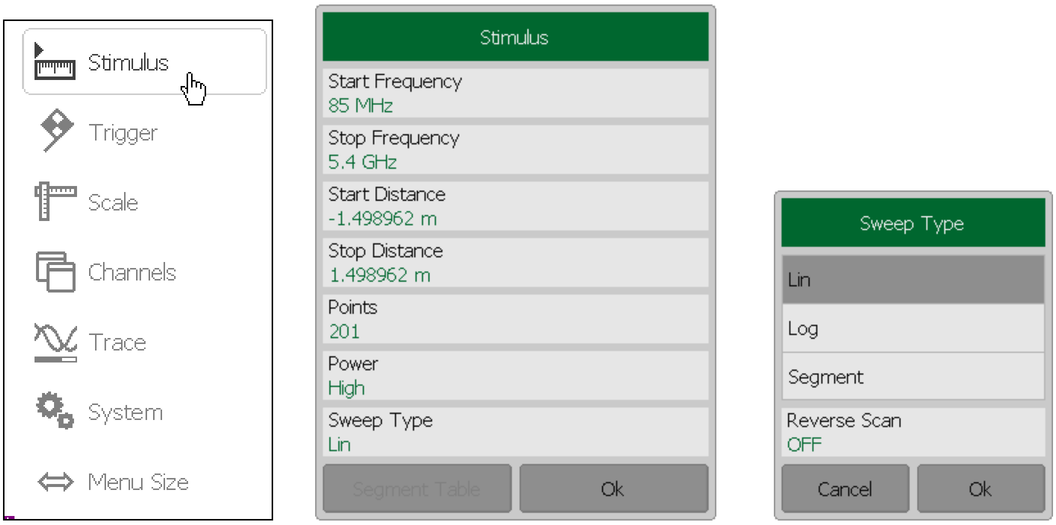
Then select the **Phase Offset** field and enter the required value using the on-screen keypad.

4.8 Stimulus Setting

The stimulus parameters are set for each channel. Before you set the stimulus parameters of a channel, make this channel active.

4.8.1 Sweep Type Setting

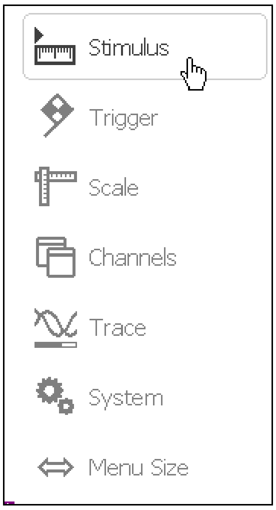
To set the sweep type use the following softkey in the right menu bar **Stimulus.**

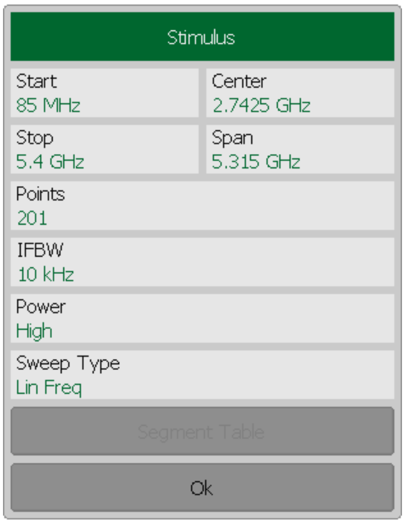


Note	If you select segment frequency sweep, the **Segment Table** softkey will be become available in **Stimulus** dialog. For segment tables details see section 4.8.5.

4.8.2 Sweep Span Setting

To enter the start and stop values of the sweep range use the following softkey in the right menu bar **Stimulus.**



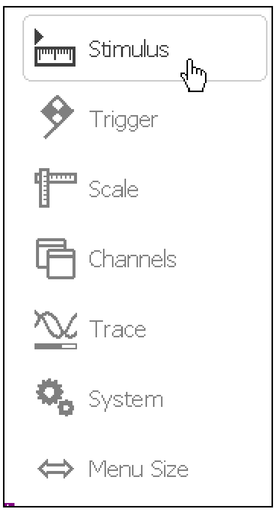


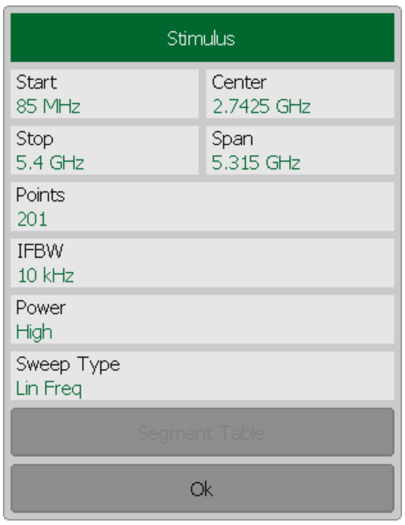
Then select the **Start Frequency** or **Stop Frequency** field and enter the required values using the on-screen keypad.

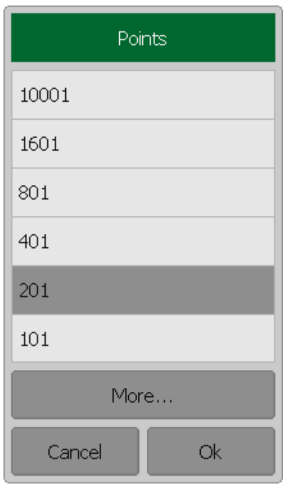
If necessary, you can select the measurement units. The current measurement units are shown to the right from the value entry field.

4.8.3 Sweep Points Setting

To enter the number of sweep points use the following softkey in the right menu bar **Stimulus.**





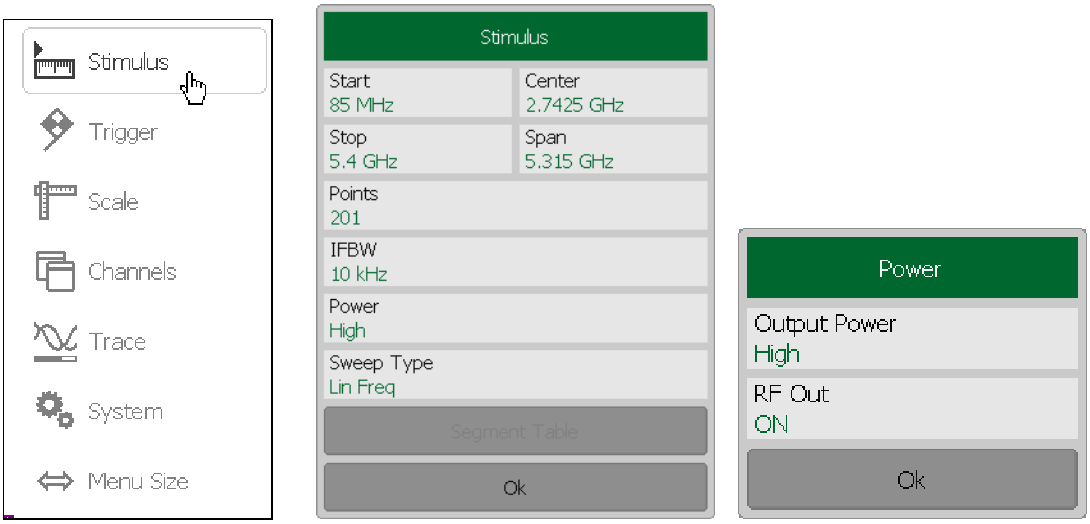


Then click on **Points** field, select the required value from the list and complete the setting by **Ok.**

4.8.4 Stimulus Power Setting

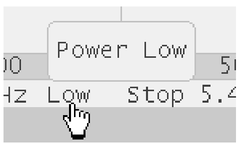
The stimulus power level can take two possible values. High output power corresponds to the source signal power of -10 dB/m. Low output power corresponds to -30 dBm.

To enter the power level value use the following softkeys in the right menu bar **Stimulus > Power.**



Then select the **Output power** field to switch between the high and low settings of the power level.

The set power level can also be seen in the channel status bar.



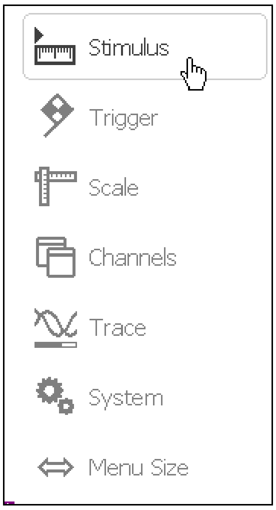
4.8.5 Segment Table Editing

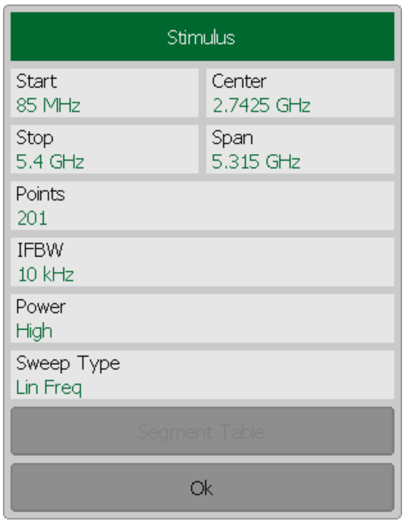
Frequency sweep span can be divided into segments. Each segment has start and stop values of the sweep range, number of points and measurement delay. IF filter and measurement delay can be enabled/disabled by the user.

The types of segment tables are shown below.

Each table line determines one segment. The table can contain one or several lines. The number of lines is limited by the aggregate number of all segment points, i.e. 100001

To edit the segment table use the following softkeys in the right menu bar **Stimulus > Segment Table.**

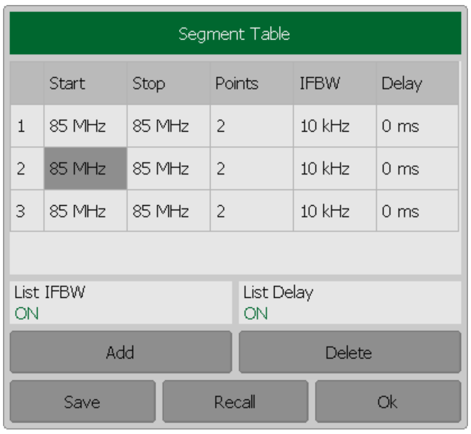




Select the segment frequency sweep to make the **Segment Table** softkey available (see section 4.8.1).

To add a segment to the segment table use **Add.**

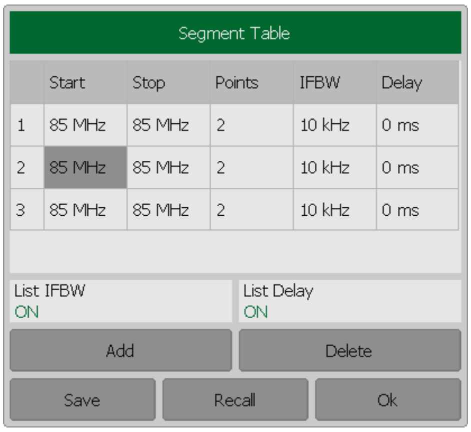
To delete a segment from the table use **Delete.**



To enter the segment parameters, move the mouse to the respective box and enter the numerical value. You can navigate the segment table using the «Up Arrow», «Down Arrow », «Left Arrow », «Right Arrow» keys.

Note The adjacent segments cannot overlap in the frequency domain.

To edit any parameter in the table, double click on the its value field and enter the required value using the on-screen keypad.



To enable/disable the IFBW filter column click on the **List IFBW** field.

To enable/disable the measurement delay column click on the **List Delay** field.

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

To save the segment table use **Save.**

To recall the segment table use **Recall.**

4.9 Trigger Setting

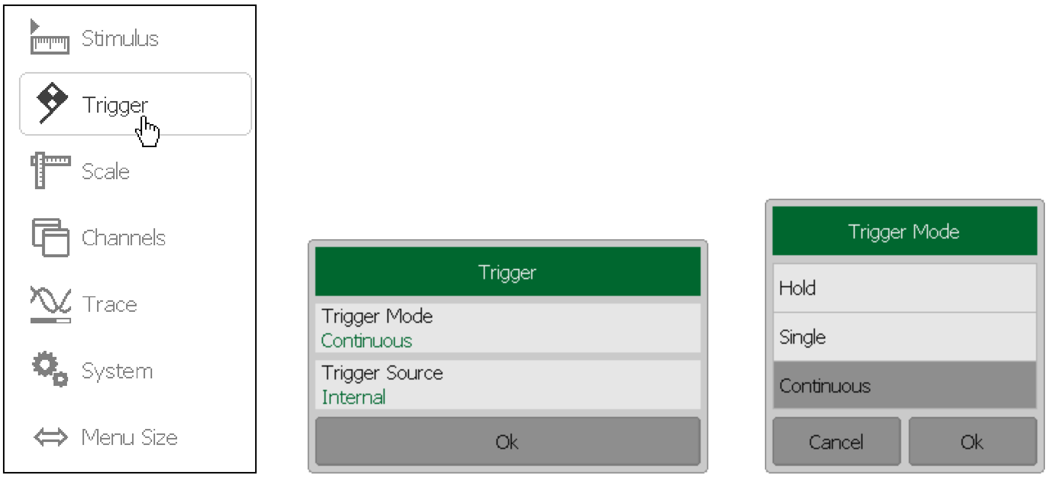
The Analyzer can operate in one of three sweep trigger modes. The trigger mode determines the sweep actuation. The trigger can have the following modes:

- Continuous – a sweep actuation occurs every time after sweep cycle is complete in each channel;
- Single – sweep actuation occurs once, and after the sweep is complete, the trigger turns to hold mode;
- Hold – sweep is stopped, the actuation does not occur.

If more than one channel window is displayed on the screen, a sweep will be actuated in them in succession.

Trigger source can be internal or bus (transferred through COM/DCOM).

To set the trigger mode use the following softkey in the right menu bar **Trigger.**



Then click on **Trigger Mode** field select the required mode from the list and complete the setting by **Ok.**

Close the Trigger dialog by **Ok.**

If you select **Single Trigger Mode** you can actuate sweep by clicking on the **Trigger Event** softkey in the right menu bar.

4.10 Measurement Optimizing

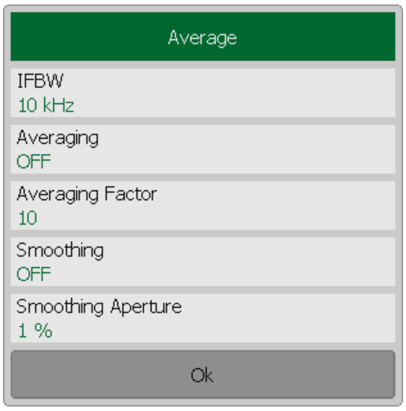
4.10.1 IF Bandwidth Setting

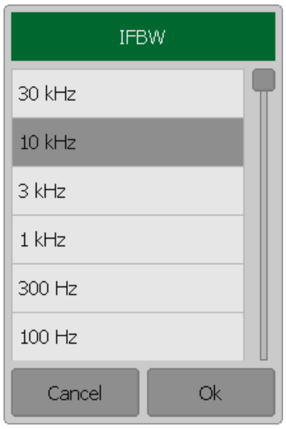
The IF bandwidth function allows the user to define the bandwidth of the test receiver. The IF bandwidth can be selected by user from the following values: 100 Hz, 300 Hz, 1 kHz, 3 kHz, 10 kHz and 30 kHz.

The IF bandwidth narrowing allows you to reduce self-noise and widen the dynamic range of the Analyzer. Also the sweep time will increase. Narrowing of the IF bandwidth to 10 will reduce the receiver noise to 10 dB.

The IF bandwidth should be set for each channel individually. Before you set the IF bandwidth, first activate the channel.

To set the IF bandwidth use the following softkey in the left menu bar **Average.**





To set the IF bandwidth click on **IFBW** field and select the required value from the list. Complete the setting by **Ok.**

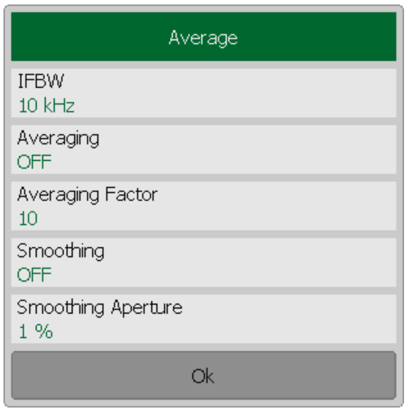
4.10.2 Averaging Setting

The averaging function is similar to IF bandwidth narrowing, it allows reducing self-noise and widening the dynamic range of the Analyzer.

The averaging in each measurement point is made over several sweeps according to the exponential window method.

The averaging should be set for each channel individually. Before you set the averaging, first activate the channel.

To set the averaging use the following softkey in the left menu bar **Average.**



To toggle the averaging function on/off click on **Average** field.

To set the averaging factor click on **Averaging Factor** field and enter the required value using the on-screen keypad.

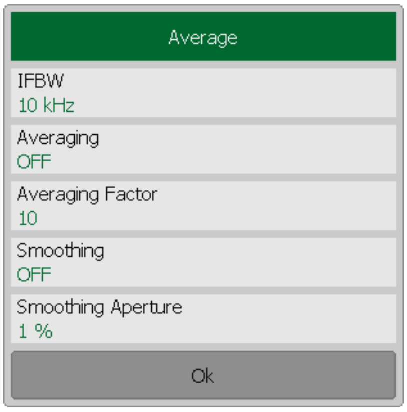
4.10.3 Smoothing Setting

The smoothing of the sweep results is made by averaging the measurement results of adjacent points of the trace determined by the moving aperture. The aperture is set by the user in percent from the total number of the trace points.

The smoothing does not increase the dynamic range of the Analyzer. It preserves the average level of the trace and reduces the noise bursts.

The smoothing should be set for each trace individually. To set the smoothing, first activate the trace.

To set the smoothing use the following softkey in the left menu bar **Average.**



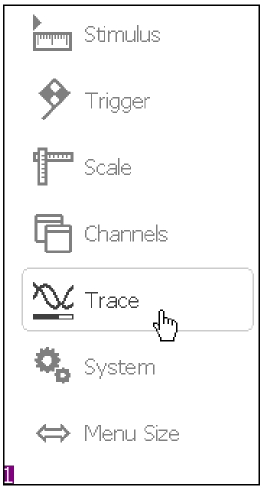
To toggle the smoothing function on/off click on **Smoothing** field.

To set the smoothing aperture click on **Smoothing Aperture** field and enter the required value using the on-screen keypad.

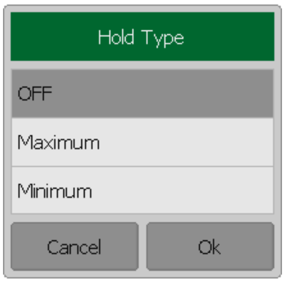
4.10.4 Trace Hold Function

The Trace Hold function displays the maximum or the minimum of any given active measurement instead the real-time data. The held data is displayed as an active trace.

To toggle the Trace Hold function on/off use the following softkeys in the right menu bar **Trace > Trace Hold.**



Then select the required type (**Maximum | Minimum**) from the **Hold Type** list and complete the setting by **Ok**.



4.11 Cable Specifications

By default, the program does NOT compensate DTF measurements to account for the inherent loss of a cable. However, to make more accurate DTF measurements, the cable loss and velocity factor can be entered using one of the following methods:

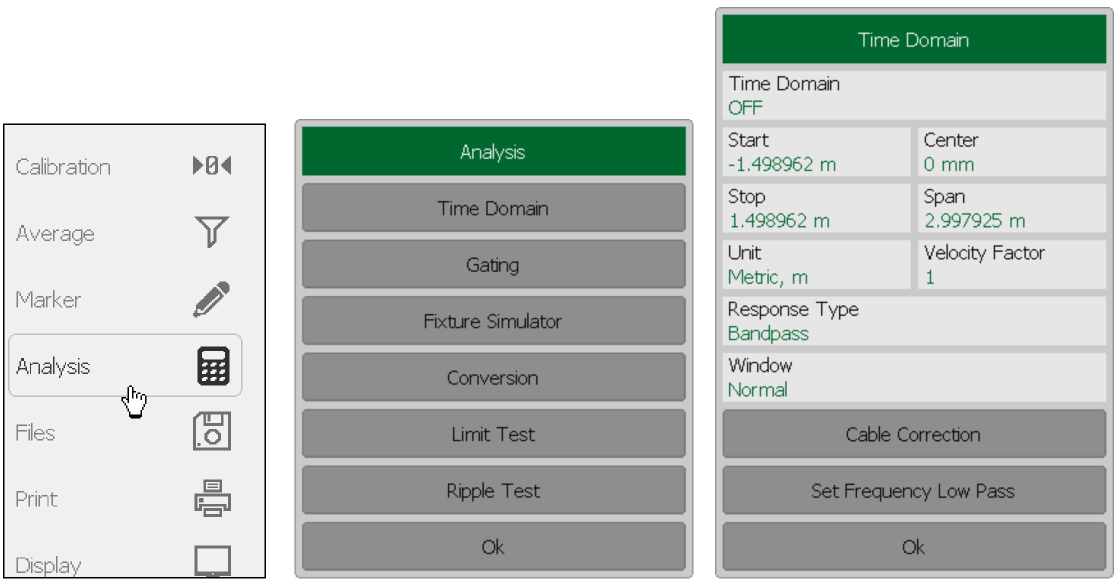
- Select a cable type from a list which contains the Cable loss in dB/meter and Velocity factor;
- Manually enter Cable loss and Velocity factor for the measurement.

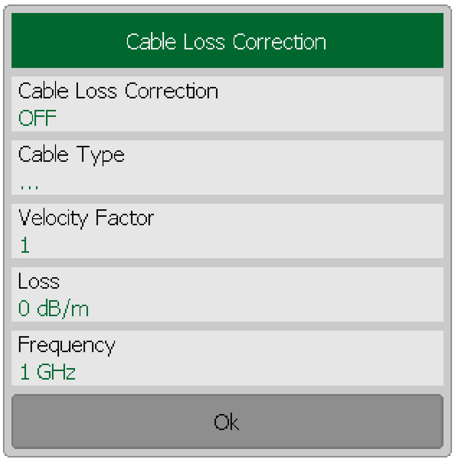
Velocity factor is a property of the physical material of a cable. A VF of 1.0 corresponds to the speed of light in a vacuum, or the fastest VF possible. A polyethylene dielectric cable has VF = 0.66 and a cable with Teflon dielectric has VF = 0.7.

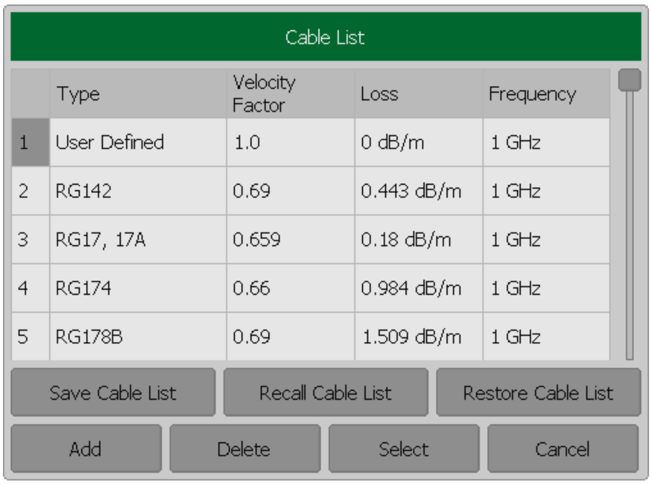
Cable Loss is specified in dB/meter. In addition to the length of the cable, loss is also directly proportional to the frequency of the signal that passes through the cable.

4.11.1 Selecting the type of cable

To select the type of cable use the following softkeys in the left menu bar **Analysis > Time Domain > Cable Correction > Cable type.**



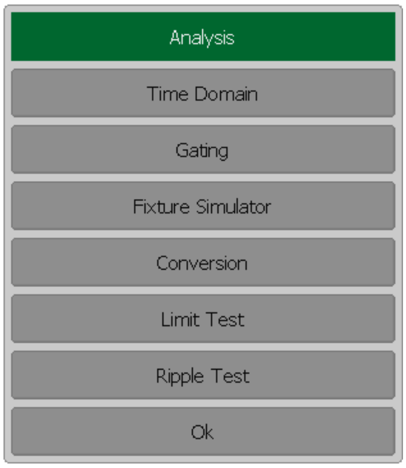


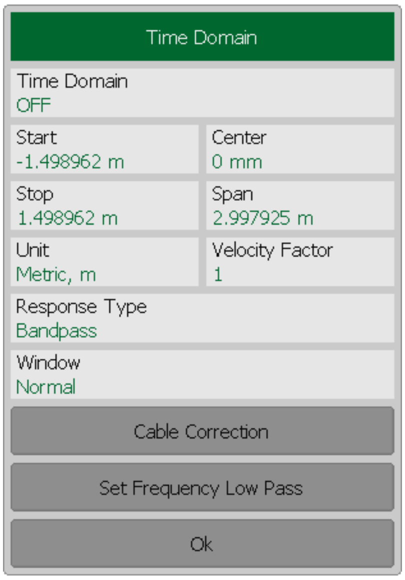


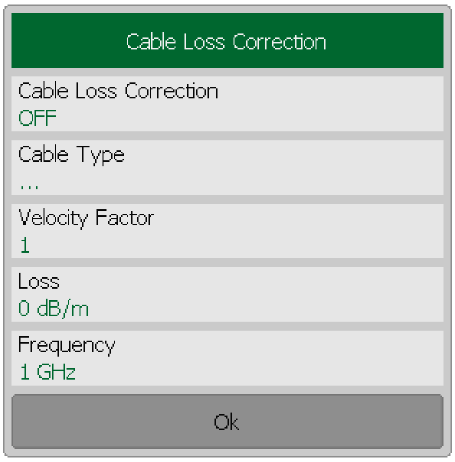
Select the required item from the **Cable List** and complete the setting by **Ok**

4.11.2 Manually specify Velocity Factor and Cable Loss

To set the parameters of cable, press the following softkeys in the left menu bar **Analysis > Time Domain > Cable Correction.**





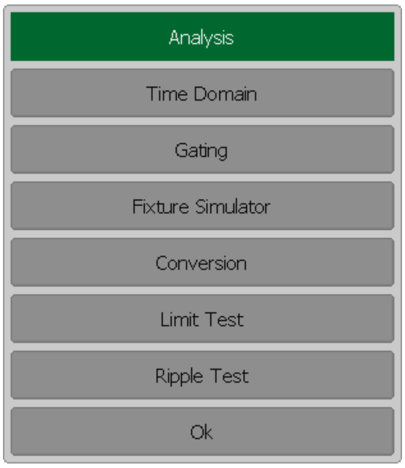


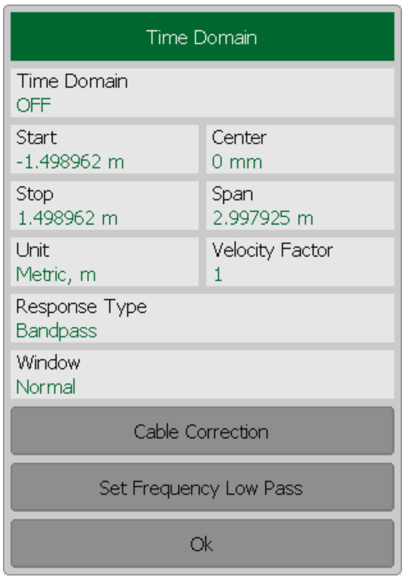
Click on **Velocity Factor** field to enter the value of velocity factor using the on-screen keypad.

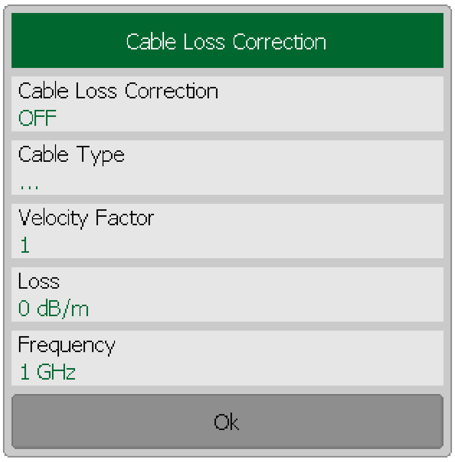
Click on **Loss** field to enter the value of cable loss using the on-screen keypad.

4.11.3 Editing table of cables

To edit the table of cables, press the following softkeys in the left menu bar **Analysis > Time Domain > Cable Correction.**







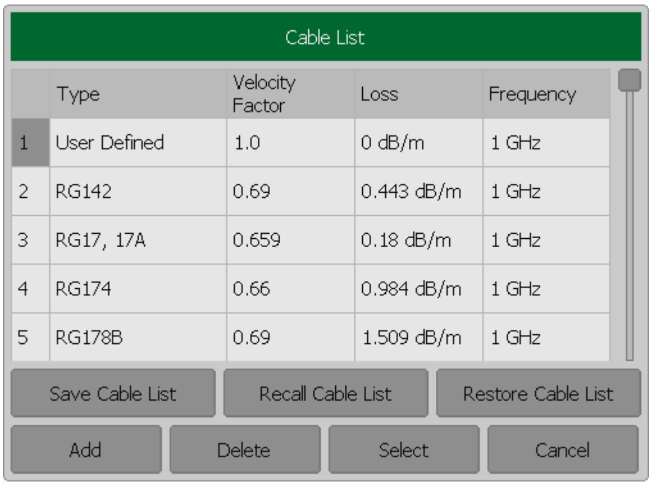
Click the left button of the mouse on the field **Cable Type.**

To add/delete rows in the table click **Add/Delete.**

Then select the required parameter in the table and double click on the corresponding cell.

Enter the required value **Cable Name, Velocity, Cable Loss** etc using the on-screen keypad.

To save the table of cables on the drive click the **Save Cable List** button.



To restore the table cables from the drive, press the **Restore Cable List** softkey.

---

5. CALIBRATION AND CALIBRATION KIT

5.1 General Information

5.1.1 Measurement Errors

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

- systematic errors,
- random errors.

Random errors comprise such errors as noise fluctuations and thermal drift in electronic components, changes in the mechanical dimensions of connectors subject to temperature drift, repeatability of connections. Random errors are unpredictable and hence cannot be estimated and eliminated in calibration. Random errors can be reduced by correct setting of the source power, IF bandwidth narrowing, maintaining constant environment temperature, observance of the Analyzer warm-up time, careful connector handling, avoidance of cable bending after calibration, and use of the calibrated torque wrench for connection of the Male-Female coaxial RF connectors.

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

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

The process of measurement of precision devices with predefined parameters with the purpose of determination of measurement systematic errors is called calibration, and such precision devices are called calibration standards. The most commonly used calibration standards are SHORT, OPEN, and LOAD.

The process of mathematical compensation (numerical reduction) for measurement systematic errors is called an error correction.

5.1.2 Systematic Errors

The systematic measurement errors of vector network analyzers are subdivided into the following categories according to their source:

- Directivity;
- Source match;
- Reflection tracking.

The measurement results till the procedure of error correction has been executed are called **uncorrected**.

The residual values of the measurement results after the procedure of error correction are called **effective**.

5.1.2.1 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 comes to the receiver of the reflected signal. Directivity errors do not depend on the characteristics of the DUT and usually have stronger effect in reflection measurements.

5.1.2.2 Source Match Error

A source match error (**Es**) is caused by the mismatch between the source test port and the input of the DUT. In this case part of the signal reflected by the DUT reflects at the test port and again comes into the input of the DUT. The error occurs both in reflection measurement and in transmission measurement. Source match errors depend on the relation between input impedance of the DUT and test port impedance.

Source match errors have strong effect in measurements of a DUT with poor input matching.

5.1.2.3 Reflection Tracking Error

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

5.1.3 Error Modeling

Error modeling and method of signal flow graphs are applied to analyzers for analysis of its systematic errors.

5.1.3.1 One-Port Error Model

In reflection measurement only test port of the Analyzer is used. The signal flow graph of errors for the test port is represented in Figure 5.1.

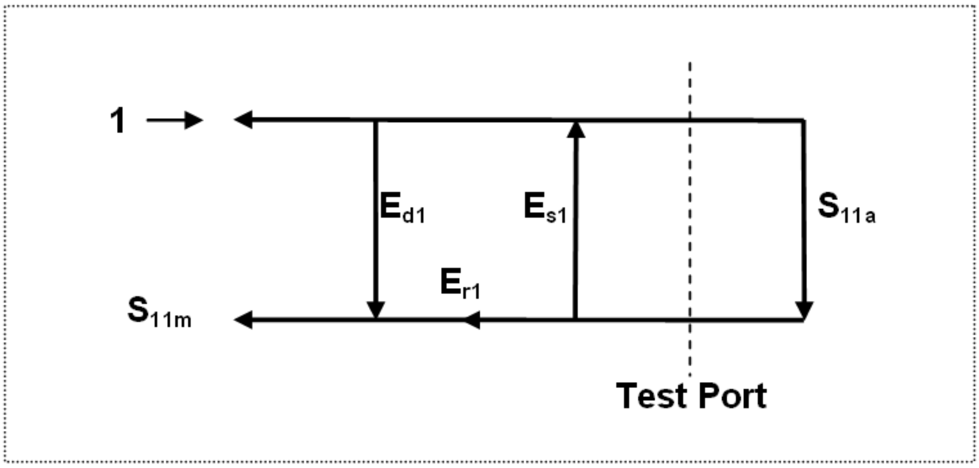


Figure 5.1 One-port error model

Where:

S_{11a} – reflection coefficient true value;
S_{11m} – reflection coefficient measured value.

The measurement result at test port is affected by the following three systematic error terms:

E_{d1} – directivity;
E_{s1} – source match;
E_{r1} – 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 **E_{d1}, E_{s1}, E_{r1}** for each measurement frequency by means of a full 1-port calibration, it is possible to calculate (mathematically subtract the errors from the measured value **S_{11m}**) the true value of the reflection coefficient **S_{11a}**.

There are simplified methods, which eliminate the effect of only one out of the three systematic errors.

5.1.4 Analyzer Test Port Defining

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

A type-N 50 Ω Male connector on the front panel of the Analyzer will be the test port if the calibration standards are connected directly to it.

Sometimes it is necessary to connect coaxial cable and/or adapter to the connector on the front panel for connection of the DUT with a different connector type. In such cases connect calibration standards to the connector of the cable or adapter.

Figure 5.2 represents two cases of test port defining for the measurement of the DUT. The use of cables and/or adapters does not affect the measurement results if they were integrated into the process of calibration.

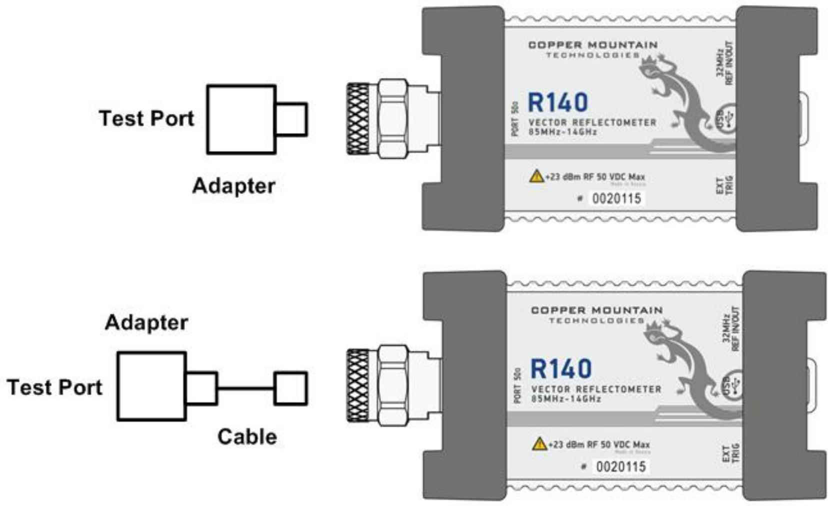


Figure 5.2 Test port defining

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

5.1.5 Calibration Steps

The process of calibration comprises the following steps:

- Selection of the calibration kit matching the connector type of the test port;
- Selection of a calibration method (see section 5.1.6) is based on the required accuracy of measurements. The calibration method determines what error terms of the model (or all of them) will be compensated;
- Measurement of the standards within a specified frequency range. The number of the 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 program and used for error correction of the measured results of any DUT.

Calibration is applied to the Analyzers channel. This means that the table of calibration coefficients is being stored for the channel.

5.1.6 Calibration Methods

The Analyzer supports several methods of calibration. The calibration methods vary by quantity and type of the standards being used, by type of error correction. The Table 5.1 represents the overview of the calibration methods.

Table 5.1 Calibration methods

Calibration Method	Parameter	Standards	Errors
Reflection Normalization	S_{11}	SHORT or OPEN	E_{r1}
Expanded Reflection Normalization	S_{11}	SHORT or OPEN LOAD	E_{r1}, E_{d1}
Full One-Port Calibration	S_{11}	SHORT OPEN LOAD	E_{r1}, E_{d1}, E_{s1}

5.1.6.1 Normalization

Normalization is the simplest method of calibration as it involves measurement of only one calibration standard for a measured S-parameter.

1-port (reflection) S-parameter (S_{11}) is calibrated by means of a SHORT or an OPEN standard, estimating reflection tracking error term **Er**.

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.

Normalization eliminates frequency-dependent attenuation and phase offset in the measurement circuit, but does not compensate errors of directivity and mismatch.

5.1.6.2 Expanded Normalization

Expanded normalization involves connection of the following two standards to the test port:

- **SHORT** or **OPEN**, and
- **LOAD**.

Measurement of the two standards allows for estimation of the reflection tracking error term **Er** and directivity error term – **Ed**.

5.1.6.3 Full One-Port Calibration

Full one-port calibration involves connection of the following three standards to the test port:

- **SHORT,**
- **OPEN,**
- **LOAD.**

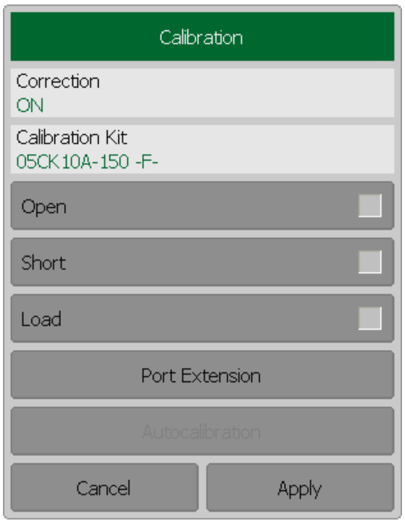
Measurement of the three standards allows for acquisition of all the three error terms **(Ed, Es,** and **Er)** of a one-port model.

5.1.6.4 Waveguide Calibration

General use and features:

- System Z0 should be set to 1 ohm before calibration. Offset Z0 and terminal impedance in the calibration standard definition also should be set to 1 ohm.
- Waveguide calibration uses two offset short standards instead of a combination of short and open standards. Typically $1/8\lambda_0$ and $3/8\lambda_0$ offset sort standards are used, where λ_0 – wave length in waveguide at the mean frequency.

In waveguide calibration, one of two offset short standards must be assigned to the open class. Consequently the GUI will contain an **Open** button with the label of this short standard.



5.1.7 Calibration Standards and Calibration Kits

Calibration standards are precision physical devices used for determination of errors in a measurement system.

A calibration kit is a set of calibration standards with a specific type of connector and specific impedance. Calibration kit includes standards of the three following types: **SHORT**, **OPEN**, and **LOAD**.

The characteristics of real calibration standards have deviations from the ideal values. For example, the ideal **SHORT** standard must have reflection coefficient magnitude equal to 1.0 and reflection coefficient phase equal to 180° over the whole frequency range. A real **SHORT** standard has deviations from these values depending on the frequency. To take into account such deviations a **calibration standard model** (in the form of an equivalent circuit with predefined characteristics) is used.

The Analyzer provides definitions of calibration kits produced by different manufacturers. The user can add the definitions of own calibration kits or modify the predefined kits using the Analyzer software. Calibration kits editing procedure is described in the section 5.3.

To ensure the required calibration accuracy select the calibration kit being used in the program menu. The procedure of calibration kit selection is described in section 5.2.1

5.1.7.1 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.**

5.1.7.2 Calibration Standard Model

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

One-port model is used for the standards **OPEN**, **SHORT**, and **LOAD** (see Figure 5.3).

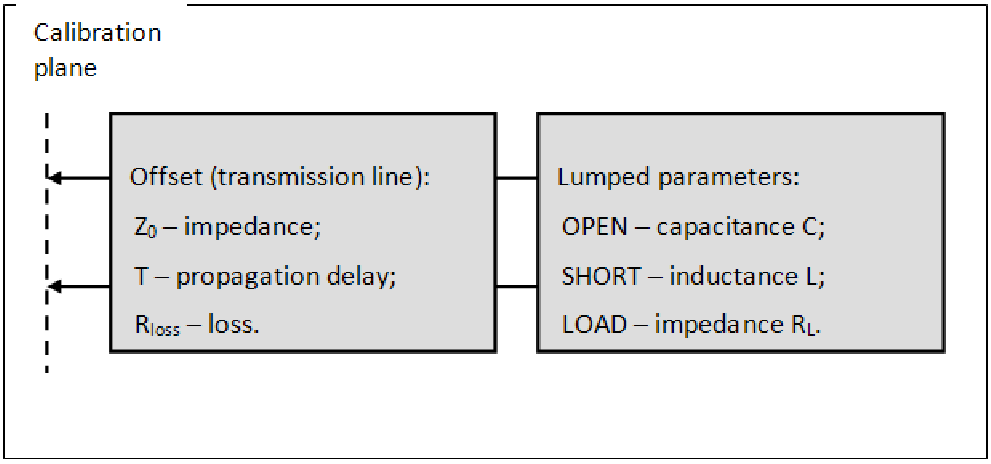


Figure 5.3 One-port standard model

The description of the numeric parameters of an equivalent circuit model of a calibration standard is shown in Table 5.2.

Table 5.2 Parameters of the calibration standard equivalent circuit model

Parameter (as in the program)	Parameter Definition
Z_0 **(Offset Z0)**	It is the offset impedance (of a transmission line) between the calibration plane and the circuit with lumped parameters.
T **(Offset Delay)**	The offset delay. It is defined as one-way propagation time (in seconds) from the calibration plane to the circuit with lumped parameters or to the other calibration plane. Each standard delay can be measured or mathematically determined by dividing the exact physical length by the propagation velocity.
R_{loss} **(Offset Loss)**	The offset loss in one-way propagation due to the skin effect. The loss is defined in [Ω/sec] at 1 GHz frequency. The loss in a transmission line is determined by measuring the delay T [sec] and loss L [dB] at 1 GHz frequency. The measured values are used in the following formula: $$Rn\left[\Omega / s\right]=\frac{L\left[dB\right]\cdot Z_0\left[\Omega\right]}{4.3429\left[dB\right]\cdot T\left[s\right]}$$
C **(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 described as a function of frequency, which is a polynomial of the third degree: $C = C_0 + C_1 f + C_2 f^2 + C_3 f^3$, where f – frequency [Hz] $C_0 \ldots C_3$ – polynomial coefficients Units: C_0[F], C_1[F/Hz], C_2[F/Hz^2], C_3[F/Hz^3]
L **(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: $L = L_0 + L_1 f + L_2 f^2 + L_3 f^3$, where f – frequency [Hz] $L_0 \ldots L_3$ – polynomial coefficients Units: L_0[H], L_1[H/Hz], L_2[H/Hz^2], L_3[H/Hz^3]

5.2 Calibration Procedures

5.2.1 Calibration Kit Selection

The Analyzer provides memory space for sixteen calibration kits. The first two items are the calibration kits with indefinite parameters. Next twelve items are the kits with manufacturer-defined parameters, available in the Analyzer by default. The other two items are the empty templates offered for user calibration kits.

The available calibration kits include the kits of Rosenberger, Agilent and Planar (see Table 5.3).

Table 5.3 Calibration kits

No.	Model Number	Calibration Kit Description
1	**Not Def 50 Ohm**	50 Ω, parameters not defined
2	**Not Def 75 Ohm**	75 Ω, parameters not defined
3	**05CK10A-150 -F-**	Rosenberger 05CK10A-150 -F- 50 Ω N-type Female, up to 18 GHz
4	**05CK10A-150 -M-**	Rosenberger 05CK10A-150 -M- 50 Ω N-type Male, up to 18 GHz
5	**N1.1 Type-N -F-**	Planar N1.1 Type-N -F- 50 Ω N-type Female, up to 1.5 GHz
6	**N1.1 Type-N -M-**	Planar N1.1 Type-N -M- 50 Ω N-type Male, up to 1.5 GHz
7	**Agilent 85032B -F-**	Agilent 85032B or 85032E, 50 Ω N-type Female, up to 6 GHz
8	**Agilent 85032B -M-**	Agilent 85032B or 85032E, 50 Ω N-type Male, up to 6 GHz
9	**Agilent 85036B -F-**	Agilent 85036B, N-type (75 Ω) Female, up to 3 GHz
10	**Agilent 85036B -M-**	Agilent 85036B, N-type (75 Ω) Male, up to 3 GHz
11	**Agilent 85032F -F-**	Agilent 85032F, 50 Ω N-type Female, up to 9 GHz

No.	Model Number	Calibration Kit Description
12	**Agilent 85032F -M-**	Agilent 85032F,50 Ω N-type Male, up to 9 GHz
13	**N611 -F-**	Copper Mountain Technologies N611 calibration kits
14	**N612 -M-**	Copper Mountain Technologies N612 calibration kits
15	**Empty**	Templates for user-defined calibration kits
16	**Empty**	Templates for user-defined calibration kits

Note -M- or -F- in the description of the kit denotes the polarity of the calibration standard connector, male or female respectively.

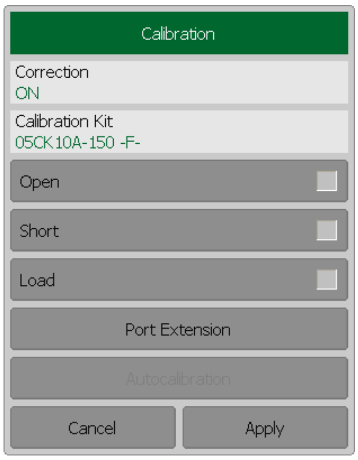
To achieve the specified measurement accuracy use a calibration kit with known characteristics.

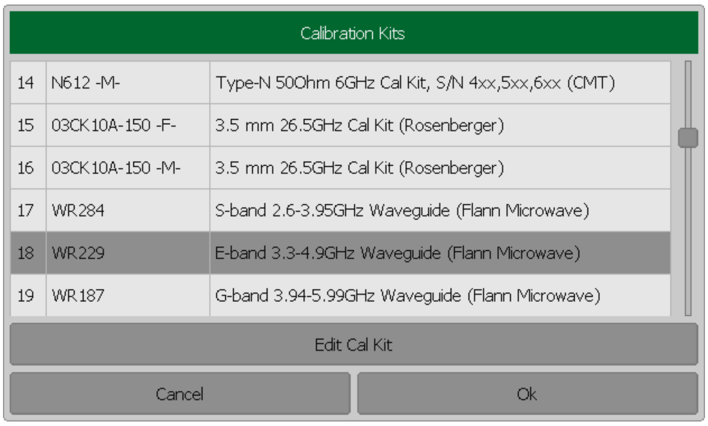
Before starting calibration select in the program the calibration kit being used among the predefined kits, or define a new one and enter its parameters.

Make sure that parameters of your calibration standards correspond to the values stored in the memory of the Analyzer. If they do not, make the required changes.

The procedure of a calibration kit definition and editing is described in section 5.3.

To select the calibration kit use the following softkey in the left menu bar **Calibration.**





The currently selected calibration kit is indicated on the softkey **Calibration Kit**, e.g. **Agilent 85032B -F-**.

Click this softkey and select the required kit from the list. Complete the setting by **Ok.**

5.2.2 Reflection Normalization

Reflection normalization is the simplest calibration method used for reflection coefficient measurement (S_{11}). Only one standard (**SHORT** or **OPEN**) is measured (see Figure 5.4) in the process of this calibration.

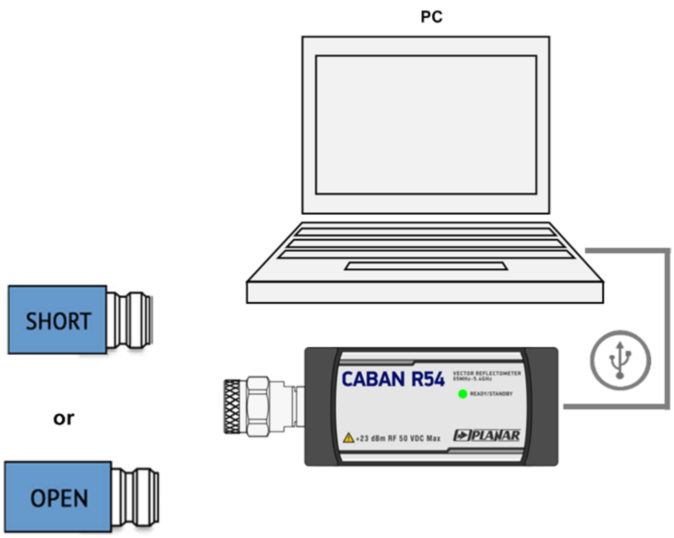
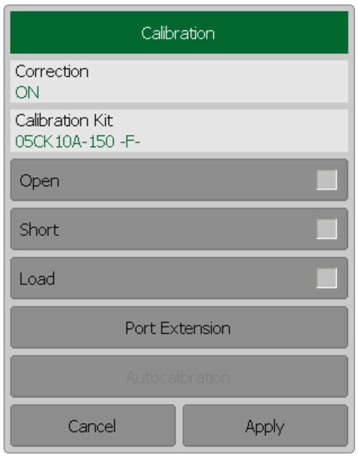


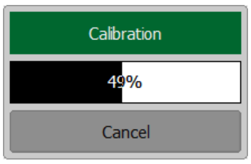
Figure 5.4 Reflection normalization

Before starting calibration perform the following settings: select active channel, set the parameters of the channel (frequency range, IF bandwidth, etc), and select the calibration kit.

To perform reflection normalization use the following softkey in the left menu bar **Calibration.**



Connect an **OPEN** or a **SHORT** standard to the test port as shown in Figure 5.4. Perform measurement using **Open** or **Short** softkey respectively.



During the measurement, a pop up window will appear in the channel window. It will have **Calibration** label and will indicate the progress of the measurement. On completion of the measurement, the left part of the **Open** or **Short** softkey will be color highlighted.

To complete the calibration procedure click **Apply.**

This will activate the process of calibration coefficient table calculation and saving it into the memory.

To clear the measurement results of the standards click **Cancel.**

This softkey does not cancel the current calibration. To disable the current calibration turn off the error correction function (see section 5.2.4).

Note	You can check the calibration status in the trace status field (see Table 5.4).

5.2.3 Full One-Port Calibration

Full one-port calibration is used for reflection coefficient measurement (S_{11}). The three calibration standards (**SHORT**, **OPEN**, and **LOAD**) are measured (see Figure 5.5) in the process of this calibration.

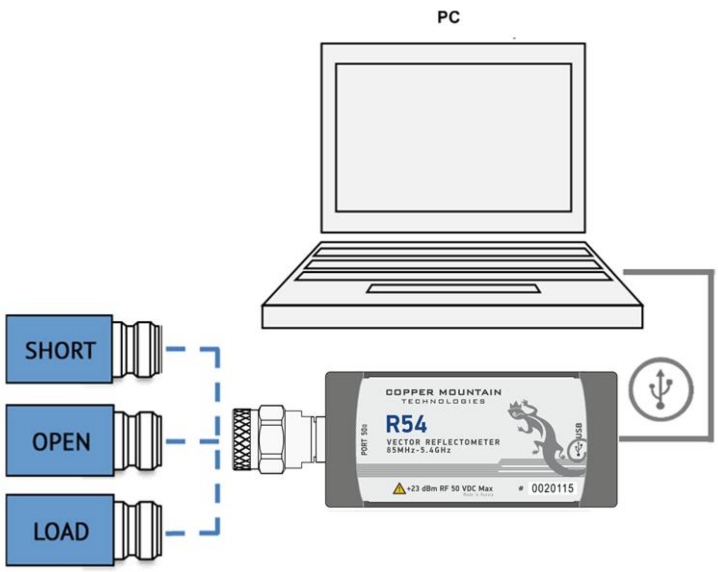
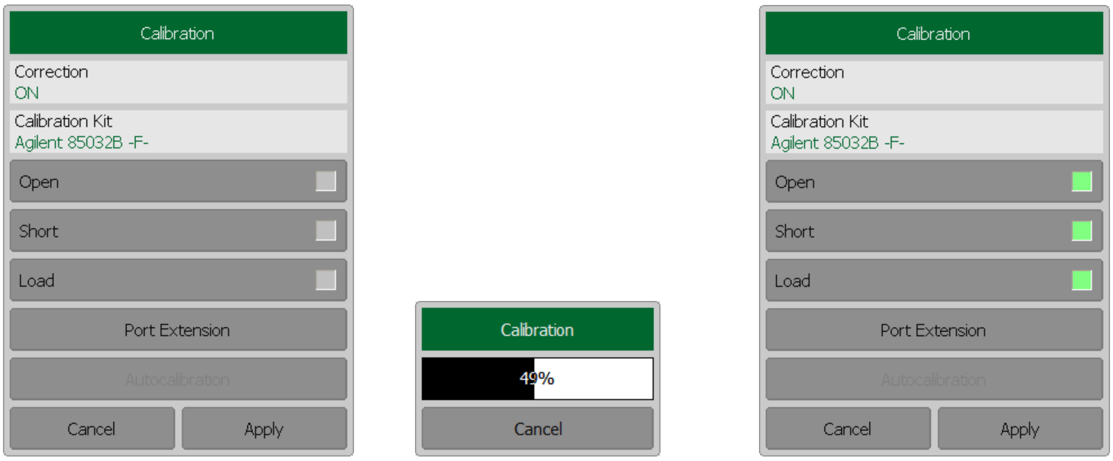


Figure 5.5 Full one-port calibration

Before starting calibration perform the following settings: select active channel, set the parameters of the channel (frequency range, IF bandwidth, etc), and select the calibration kit.

To perform full one-port calibration use the following softkey in the left menu bar **Calibration.**

Connect **SHORT**, **OPEN** and **LOAD** standards to the test port in any sequense as shown in Figure 5.5. Perform measurements clicking the softkey corresponding to the connected standard **Open, Short** or **Load** respectively.



During the measurement, a pop up window will appear in the channel window. It will have **Calibration** label and will indicate the progress of the measurement. On completion of the measurement, the left part of the **Open**, **Short** or **Load** softkey will be color highlighted.

To complete the calibration procedure click **Apply.**

This will activate the process of calibration coefficient table calculation and saving it into the memory.

To clear the measurement results of the standards click **Cancel.**

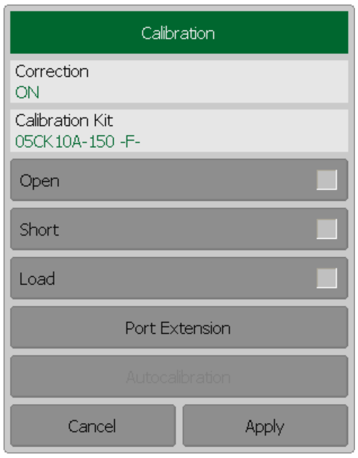
This softkey does not cancel the current calibration. To disable the current calibration turn off the error correction function (see section 5.2.4).

Note	You can check the calibration status in the trace status field (see Table 5.4).

5.2.4 Error Correction Disabling

This feature allows the user to disable the error correction function.

To disable and enable again the error correction function use the following softkey in the left menu bar **Calibration.**



Click on **Correction** field to toggle the on/off settings of the correction state. Close the dialog by **Apply.**

Note	When you turn off the error correction function, **Correction Off** message will appear in the program status bar. 

5.2.5 Error Correction Status

The error correction status for each individual trace is indicated in the trace status field (see Table 5.4). For trace status field description, see section 4.2.2.

Table 5.4 Trace error correction status

Symbols	Definition
RO	OPEN response calibration
RS	SHORT response calibration
F1	Full 1-port calibration

5.2.6 System Impedance Z0

Z_0 is the system impedance of a measurement path. Normally it is equal to the impedance of the calibration standards, which are used for calibration. The Z_0 value should be specified before calibration, as it is used for calibration coefficient calculations.

Note	Selection of calibration kit automatically determines the system impedance Z_0 in accordance with the value specified for the kit.

5.2.7 Port Extension

The port extension function enables you to eliminate the fixture (with or without losses) effects on the measurement results. The function virtually extends the test ports moving the calibration plane to the terminals of the DUT (by the length of the fixture). The fixture parameters are defined by the user for each port individually (see Figure 5.6).

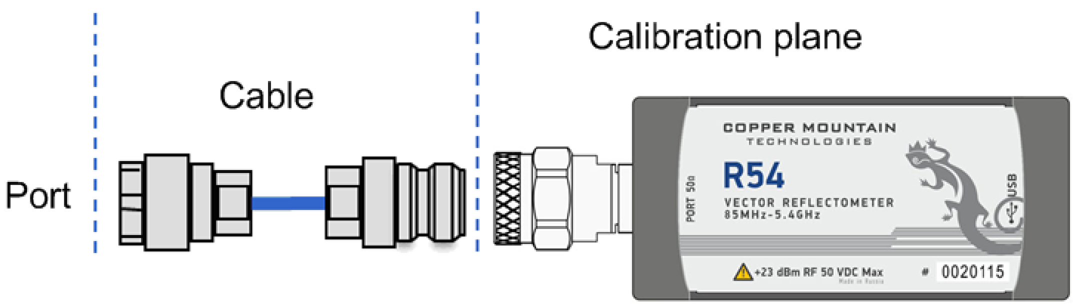


Figure 5.6 Port extension

The phase incursion caused by electrical delay is compensated for, when a lossless fixture needs to be removed:

$$e^{j\cdot 2\pi\cdot f\cdot t}$$, where

f - frequency, Hz,

t - electrical delay, sec.

The feature of removing a lossless fixture is similar to the feature of electrical delay setting for a trace (see section 4.7.7), but unlike the latter it is applied to all the traces of the channel. It compensates for a fixture length in transmission measurements and for a double fixture length in reflection measurements.

To remove a fixture with losses, the following methods of loss defining (in one, two or three frequency points) are applied:

- Frequency-independent loss at DC - $\mathbf{L_0}$

$$L(f) = L_0$$

- Frequency-dependent loss determined by the losses in two frequency points: $\mathbf{L_0}$ at DC, and $\mathbf{L_1}$ at frequency $\mathbf{F_1}$

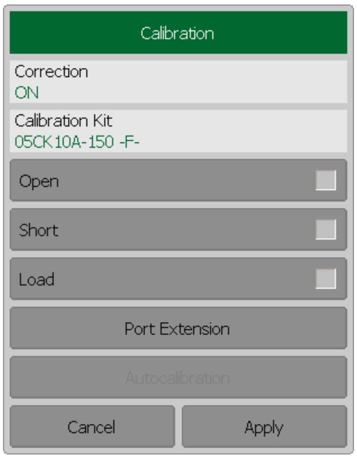
$$L(f) = L_0 + (L_1 - L_0)\cdot\sqrt{\frac{f}{F_1}}$$

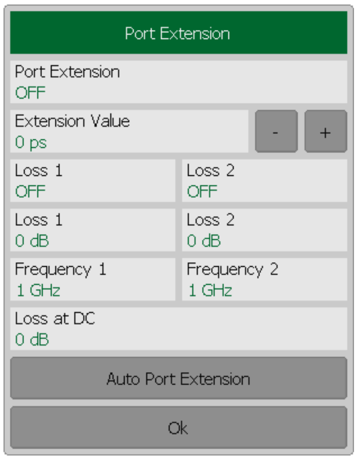
- Frequency-dependent loss determined by the losses in three frequency points: $\mathbf{L_0}$ at DC, $\mathbf{L_1}$ at frequency $\mathbf{F_1}$, and $\mathbf{L_2}$ at frequency $\mathbf{F_2}$

$$L(f) = L_0 + (L_1 - L_0)\cdot\left(\frac{f}{F_1}\right)^n,$$

$$n = \frac{\log\left|\frac{L_1}{L_2}\right|}{\log\left|\frac{F_1}{F_2}\right|}$$

To set the Port Extension use the following softkeys **Calibration > Port Extension.**





Click on **Port Extension** field to toggle the on/off settings of the Port Extension state.

Then select the **Extension Port** field and enter the required value using the on-screen keypad.

Use **Loss at DC** to determinate L_0.

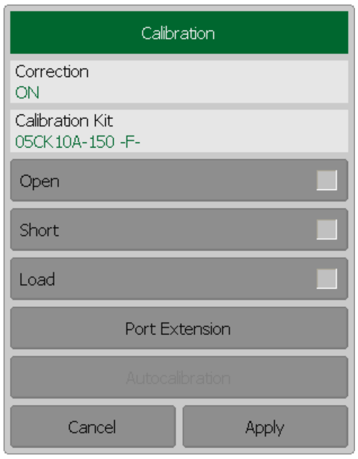
Use **Loss 1** and **Freq 1** to determinate L_1.and F_1.

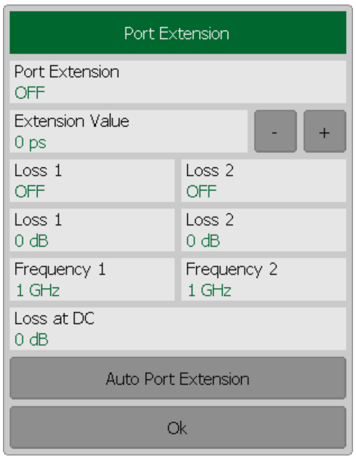
Use **Loss 2** and **Freq 2** to determinate L_2.and F_2.

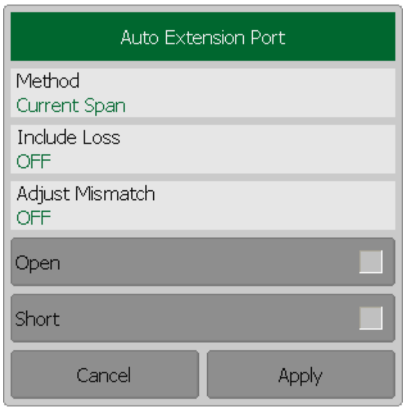
Close the dialog by **Ok.**

5.2.8 Auto Port Extension

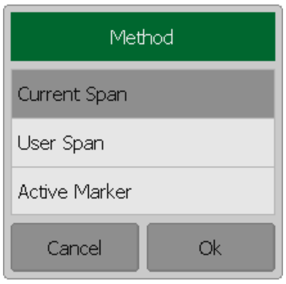
To apply the Auto Port Extension use the following softkeys **Calibration > Port Extension > Auto Port Extension.**







Click on **Method** field to select method of calculation of extension port (**Current Span, User Span** or **Active Marker**).



Click on **Include Loss** or **Adjust Mismatch** field to toggle the on/off status of this settings.

Use softkeys: **Open** or **Short** or **Open** and **Short** to execute a measurement and calculate extension of port.

Close the dialog by **Apply.**

5.3 Calibration Kit Management

This section describes how to edit the calibration kit description.

The Analyzer provides a table for 16 calibration kits. The first fourteen kits are the predefined kits. The last two kits are empty templates for adding calibration standards by the user.

A calibration kit alteration can be required to precise the standard parameters to improve the calibration accuracy.

A new user-defined calibration kit can be added when a required kit is not included in the list of the predefined kits.

The changes made by the user to the definition of the calibration kits are saved into the calibration kit configuration file in the program working folder. For the saving no additional manipulations are required.

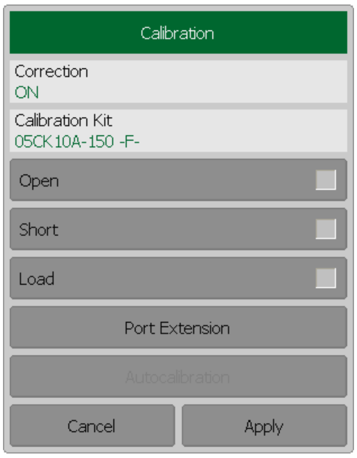
Note	Changes to a predefined calibration kit can be cancelled any time and the initial state will be restored by the **Restore** softkey in **Calibration Kit Editor** dialog.

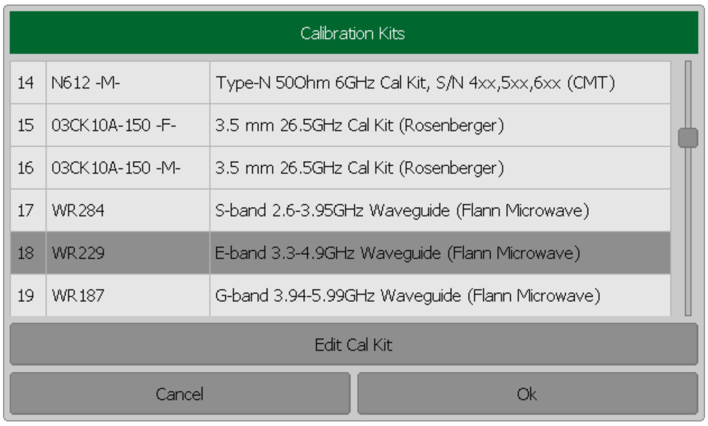
5.3.1 Calibration Kit Selection for Editing

The calibration kit currently selected for calibration is the kit available for editing. This active calibration kit is selected by the user as described in section 5.2.1.

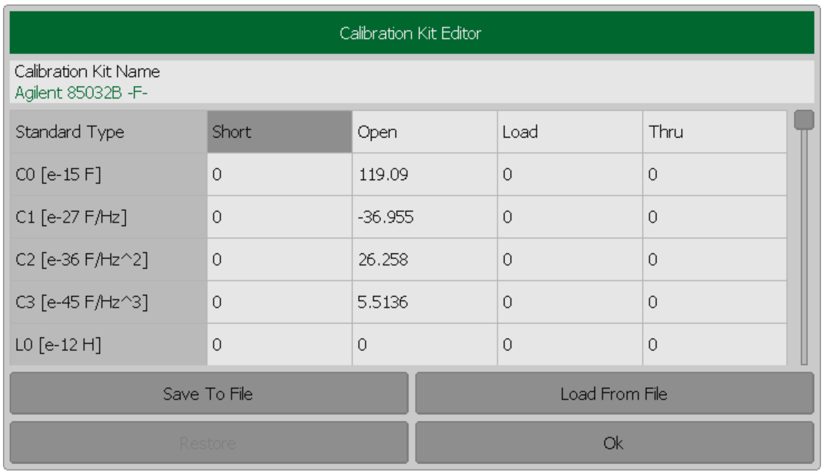
5.3.2 Calibration Kit Label Editing

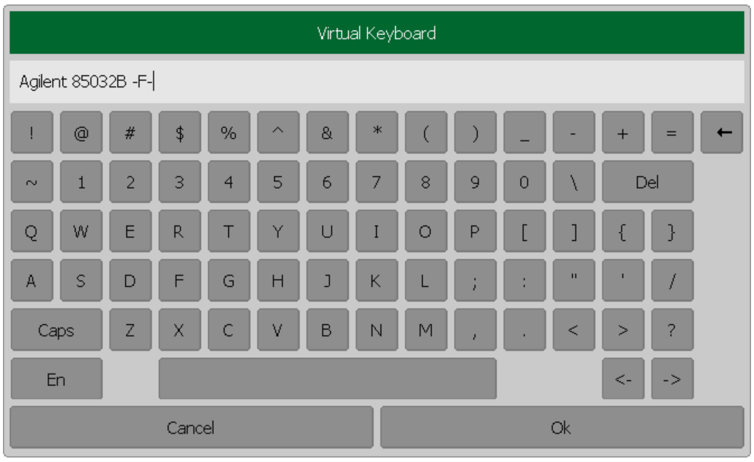
To edit the label of a calibration kit use the following softkeys in the left menu bar **Calibration > Calibration Kit > Edit Cal Kit.**





Click on **Calibration Kit Name** field and enter the calibration kit label using the on-screen keypad.

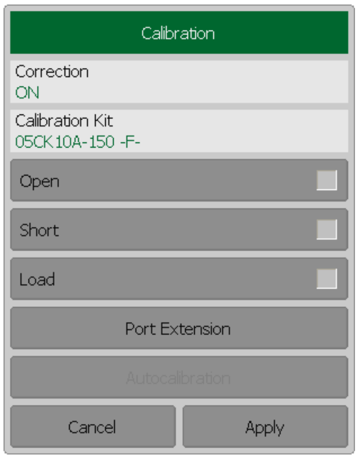


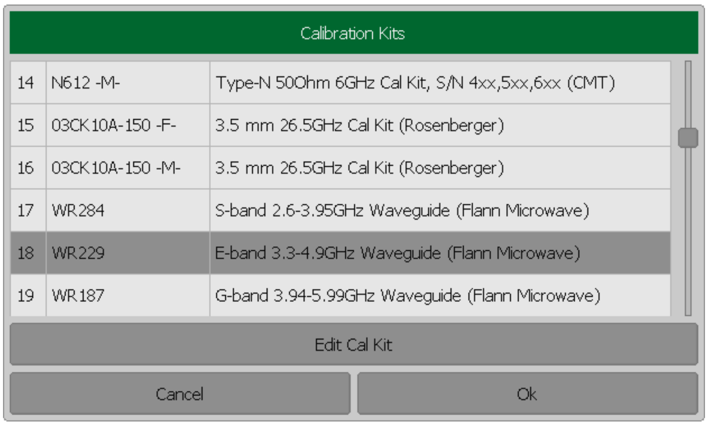


To save the settings and close the dialog click **Ok.**

5.3.3 Predefined Calibration Kit Restoration

To cancel the user changes of a predefined calibration kit use the following softkey **Calibration > Calibration Kit.**

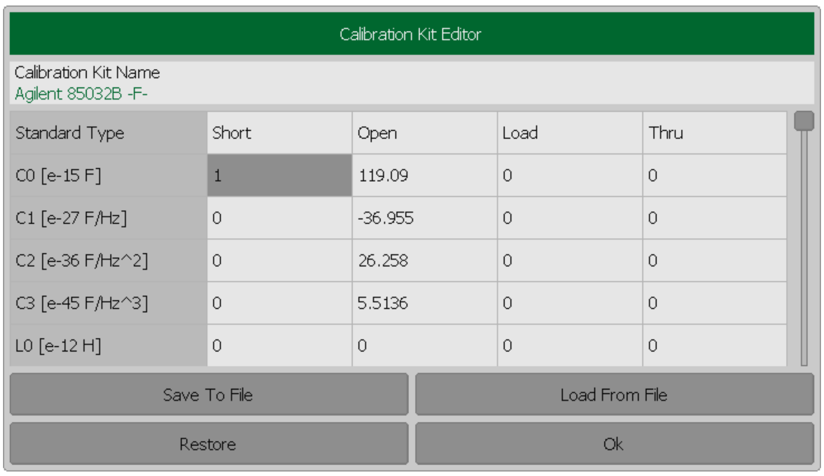




Select the required kit from the list and click **Edit Cal Kit.**

If the kit parameters differ from the predefined ones, **Restore** softkey becomes available.

To cancel your changes click **Restore.**

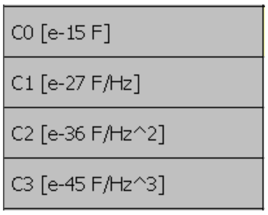


Close the dialog by **Ok.**

5.3.4 Calibration Standard Editing

To edit the calibration standard parameters use the following softkeys **Calibration > Calibration Kit > Edit Cal Kit.**

Then select the required parameter in the table and double click on the corresponding cell. Enter the required value using the on-screen keypad.

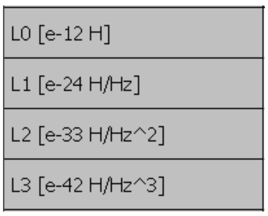


For an OPEN standard, the values of the fringe capacitance of the OPEN model are specified. This model is described by the following polynomial of the third order:

$C = C_0 + C_1 f + C_2 f^2 + C_3 f^3$, where

f: frequency [Hz]

$C_0 \ldots C_3$ – polynomial coefficients

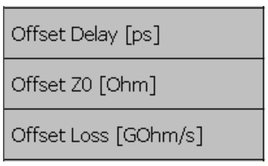


For a SHORT standard, the values of the residual inductance of the SHORT model are specified. This model is described by the following polynomial of the third order:

$L = L_0 + L_1 f + L_2 f^2 + L_3 f^3$, where

f : frequency [Hz]

$L_0 \ldots L_3$ – polynomial coefficients



The parameters of the transmission line of the standard model are specified for all the types of the standards.

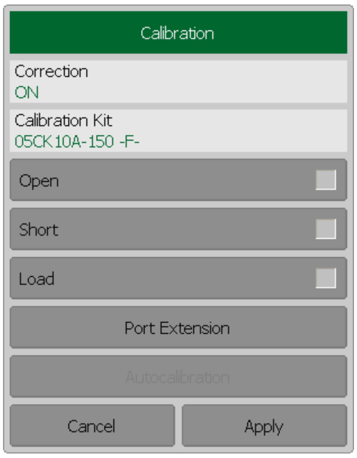
- Offset delay value in one direction (s);

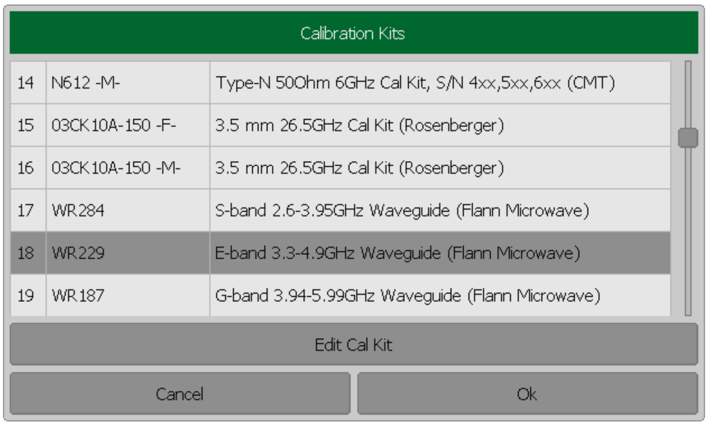
	• Offset wave impedance value (Ω); • Offset loss value (Ω/s).

5.3.5 Calibration Standard Defining by S-Parameter File

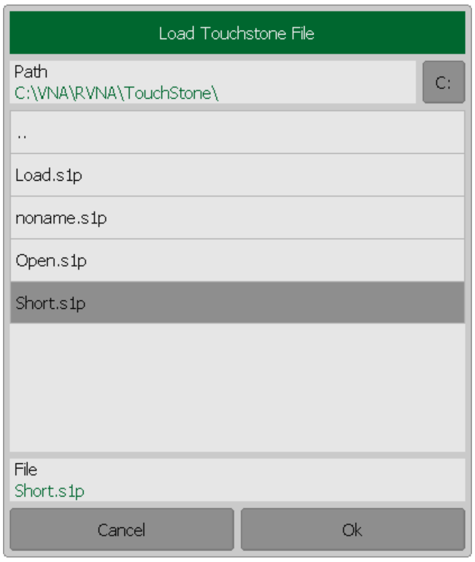
Parameters of a calibration standard can be set from an S-parameter file in Touchstone format.

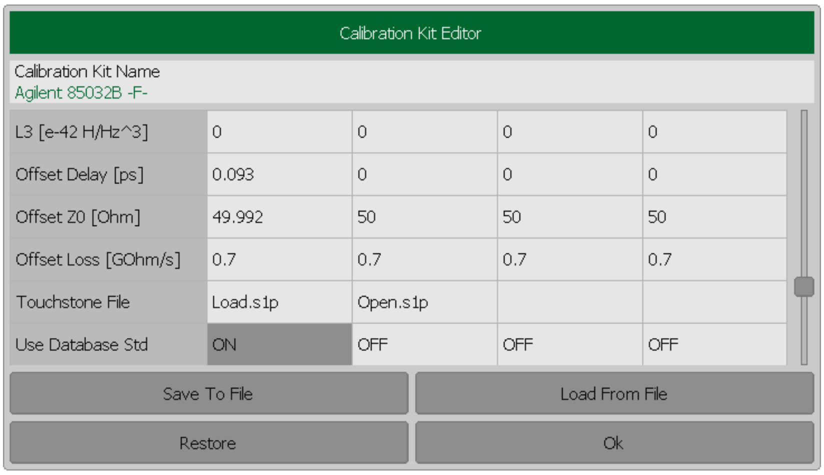
To set the calibration standard parameters by S-parameter file use the following softkeys **Calibration > Calibration Kit > Edit Cal Kit.**





In the **Calibration Kit Editor** dialog select the **Touchstone file** row. Then select the cell with the required standard and double click on it. Dialog for file selection will appear.





Select **Use Database Std** row in the table and the cell with the required standard type. Double click on the cell will toggle the on/off status.

Note	If a file in the Touchstone format is not uploaded or its format is improper, it will be impossible to use the S-parameter file to define the calibration standard.

5.4 Automatic Calibration Module

Automatic calibration module (ACM) is a special device, which allows automating of the process of calibration. ACM is shown in Figure 5.7.

Figure 5.7 Automatic Calibration Module

ACM offers the following advantages over the traditional SOLT calibration, which uses a mechanical calibration kit:

- Reduces the number of connections of standards. Instead of connecting seven standards, it requires connecting only two ACM connectors;
- Reduces the calibration time;
- Reduces human error probability;
- Potentially provides higher accuracy.

ACM has two RF connectors for connection to the Analyzer test ports and a USB connector for control. ACM contains electronic switches, which switch between different reflection and transmission impedance states, as well as memory, which stores precise S-parameters of these impedance states.

After you connect the ACM to the Analyzer, the Analyzer software performs the calibration procedure automatically, i.e. switches between different ACM states, measures them, and computes calibration coefficients using the data stored in the ACM memory.

5.4.1 Automatic Calibration Module Features

Calibration Types:

ACM allows the Analyzer software to perform 1-Path two-port or full one-port calibrations with the click of a button. We recommend you to terminate the unusable ACM port with a load while performing one-port calibration.

Characterization:

Characterization is a table of S-parameters of all the states of the ACM switches, stored in the ACM memory. There are two types of characterization: user characterization and factory characterization. ACM has two memory sections. The first one is write-protected and contains factory characterization. The second memory section allows you to store up to three user characterizations. Before calibration you can select the factory characterization or any of the user characterizations stored in the ACM memory. The user characterization option is provided for saving new S-parameters of the ACM after connecting adapters to the ACM ports.

Automatic Orientation:

Orientation means relating the ACM ports to the test ports of the Analyzer. While the Analyzer test ports are indicated by numbers, the ACM ports are indicated by letters A and B.

Orientation is defined either manually by the user, or automatically. The user is to select the manual or automatic orientation method. In case of automatic orientation, the Analyzer software determines the ACM orientation each time prior to its calibration or characterization.

Thermal Compensation:

The most accurate calibration can be achieved if the ACM temperature is equal to the temperature, at which it was characterized. When this temperature changes, certain ACM state parameters may deviate from the parameters stored in the memory. This results in reduction of the ACM calibration accuracy.

To compensate for the thermal error, the ACM has thermal compensation function. Thermal compensation is a software function of the ACM S-parameter correction based on its temperature dependence and the data from the temperature sensor inside the ACM. The temperature dependence of each ACM is determined at the factory and saved into its memory.

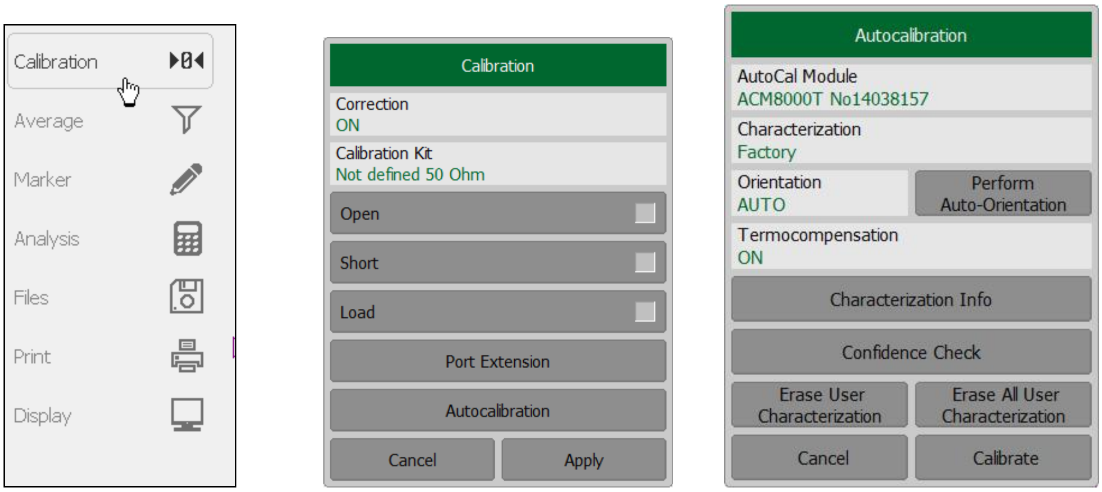
The function of thermal compensation can be enabled or disabled by the user.

5.4.2 Automatic Calibration Procedure

Before calibrating the Analyzer with ACM, perform some settings, i.e. activate a channel and set channel parameters (frequency range, IF bandwidth, etc).

Connect the ACM to the Analyzer test ports, and connect the USB port of the ACM to the USB port of the computer.

To start automatic calibration use the following softkeys **Calibration > Autocalibration > Calibrate.**



Select manual or automatic orientation of the ACM using **Orientation** field. It is recommended to select **AUTO** orientation.

Enable or disable the thermal compensation using **Thermocompensation** field.

To display detailed information on characterization use **Characterization Info** softkey.

---

6. MEASUREMENT DATA ANALYSIS

6.1 Markers

A marker is a tool for numerical readout of a stimulus value and a measured parameter value in a specific point on the trace. You can activate up to 16 markers on each trace. See a trace with two markers in Figure 6.1.

The markers allow the user 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 minimum, maximum, peak and pre-defined values on the trace;
- Determining trace parameters (statistics, bandwidth, etc).

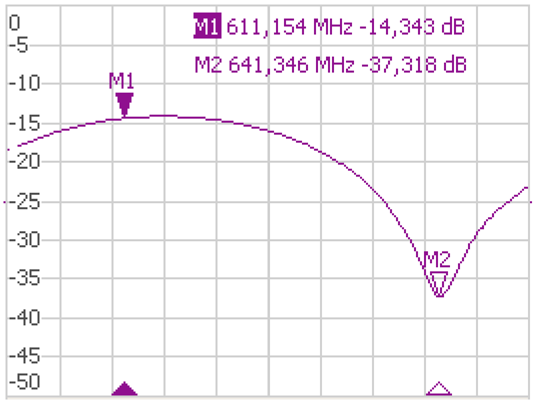


Figure 6.1

Markers can have the following indicators:

M1 ▼	symbol and number of the active marker on a trace,
M2 ▽	symbol and number of the inactive marker on a trace,
▲	symbol of the active marker on a stimulus axis,
△	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 inverse color.

The marker data field contents vary depending on the display format (rectangular or circular).

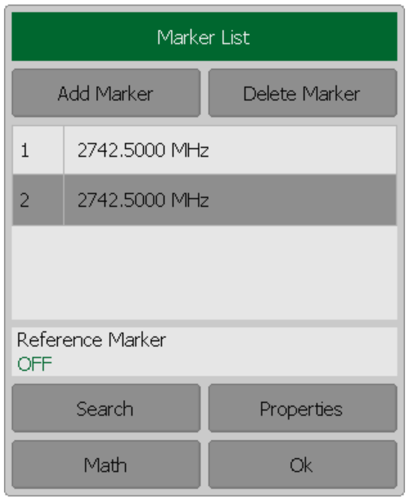
In rectangular format, the marker shows the measurement parameter value plotted along Y-axis in the active format (see Table 4.6).

In circular format, Smith chart (R+jX), the marker shows the following values:

- Resistance (Ω);
- Reactance (Ω);
- Equivalent capacitance or inductance (F/H).

6.1.1 Marker Adding

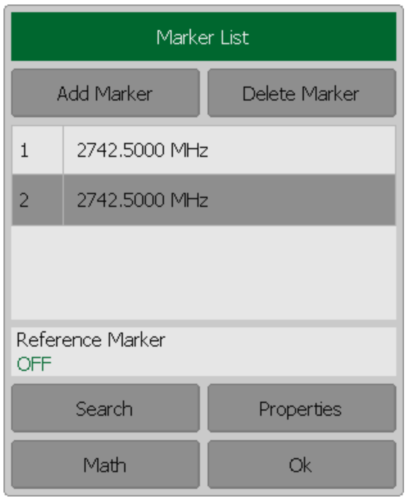
To enable a new marker use the following softkeys **Marker > Add Marker.**



Note	The new marker appears as the active marker in the middle of the stimulus axis.

6.1.2 Marker Deleting

To delete an active marker use the following softkeys **Marker > Delete Marker**.



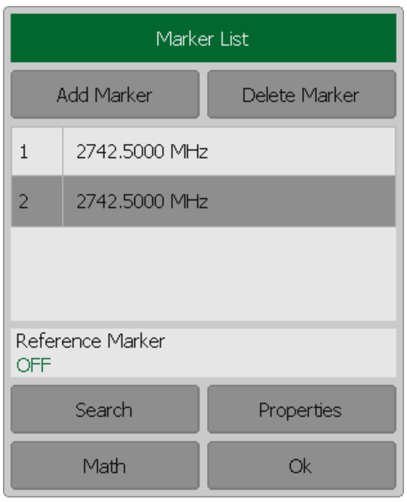
Note The active marker is highlighted in the **Marker List** dialog.

6.1.3 Marker Stimulus Value Setting

Before you set the marker stimulus value, you need to select the active marker.

You can set the stimulus value by entering the numerical value from the keyboard or by dragging the marker using the mouse. Drag-and-drop operation is described in section 4.3.6.

To set the marker stimulus value use the following softkey **Marker.**



Select a required marker from the list.

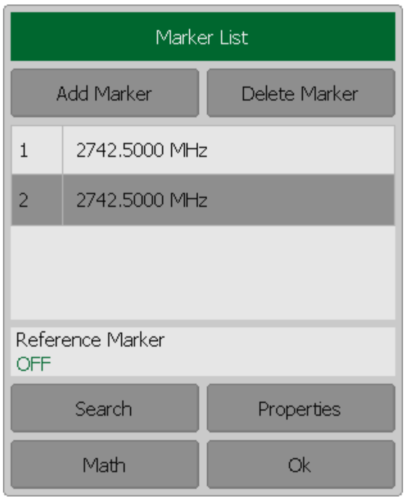
Double click on the marker stimulus value in the table, and enter the stimulus value using the on-screen keypad.

Complete the setting by **Ok.**

Note	To enter the stimulus numerical value in the marker data field, you have to click on it.

6.1.4 Marker Activating

To activate a marker use the softkey **Marker.**



In the **Marker List** dialog click on the marker number to activate it.

Note	You can activate a marker on the trace by clicking on it.

6.1.5 Reference Marker Feature

Reference marker feature allows the user to view the data relative to the reference marker. Other marker readings are represented as delta relative to the reference marker. The reference marker shows the absolute data. The reference marker is indicated with R symbol instead of a number (see Figure 6.2). Enabling of a reference marker turns all the other markers to relative display mode.

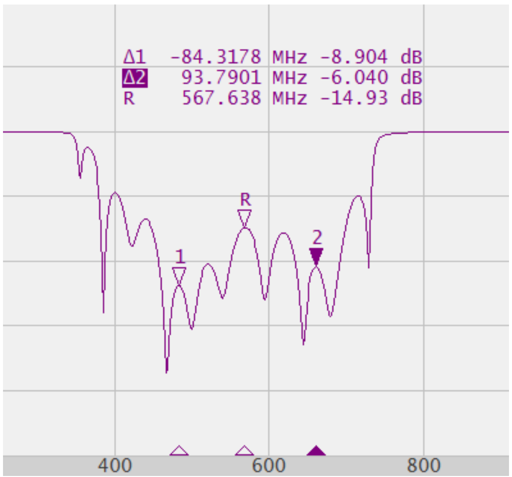


Figure 6.2

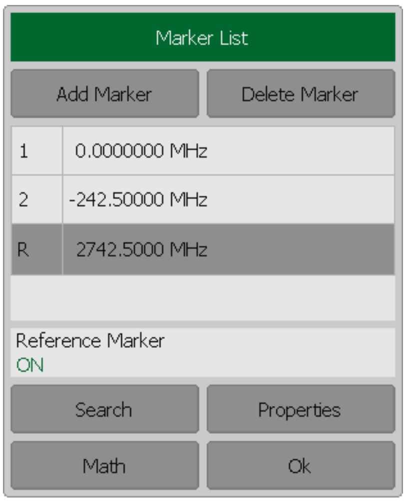
Reference marker can be indicated on the trace as follows:

R ▼	symbol of the active reference marker on a trace;
R ▽	symbol of the inactive reference marker on a trace.

The reference marker displays the stimulus and measurement absolute values. All the rest of the markers display the relative values:

- stimulus value – difference between the absolute stimulus values of this marker and the reference marker;
- measured value – difference between the absolute measurement values of this marker and the reference marker.

To enable/disable the reference marker feature use the softkey **Marker.**

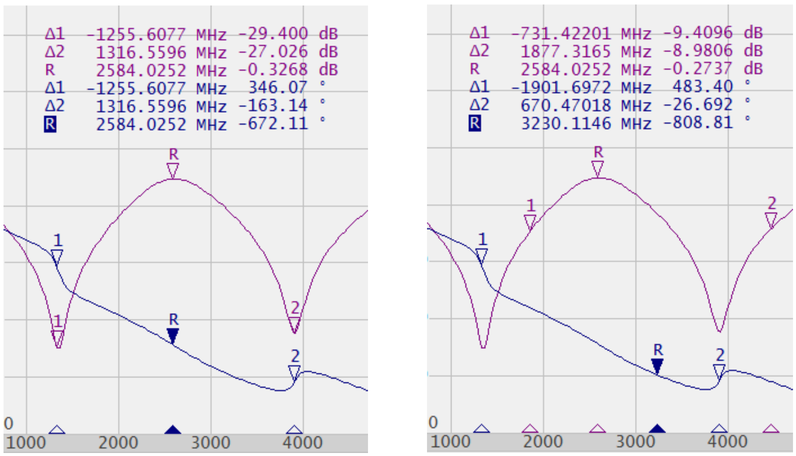


Click on the Reference Marker status field to toggle the status of the reference marker. The reference marker will be added to / deleted from the marker list and the trace.

6.1.6 Marker Properties

6.1.6.1 Marker Coupling Feature

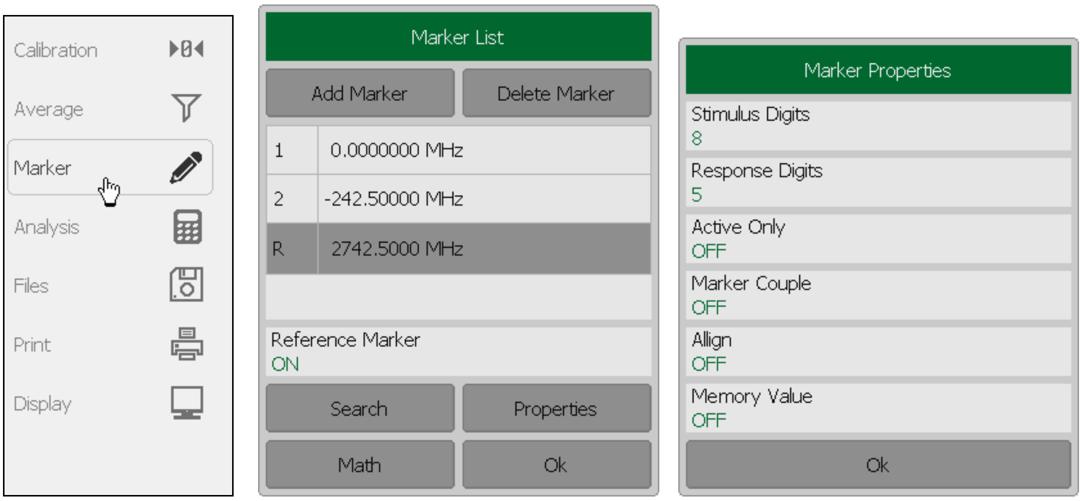
The marker coupling feature enables/disables dependence of the markers of the same numbers on different traces. If the feature is turned on, the coupled markers (markers with same numbers) will move along X-axis synchronously on all the traces. If the coupling feature is off, the position of the markers with same numbers along X-axis will be independent (see Figure 6.3).



Coupling: ON Coupling: OFF

Figure 6.3 Marker coupling feature

To enable/disable the marker coupling feature use the following softkeys **Marker > Properties.**



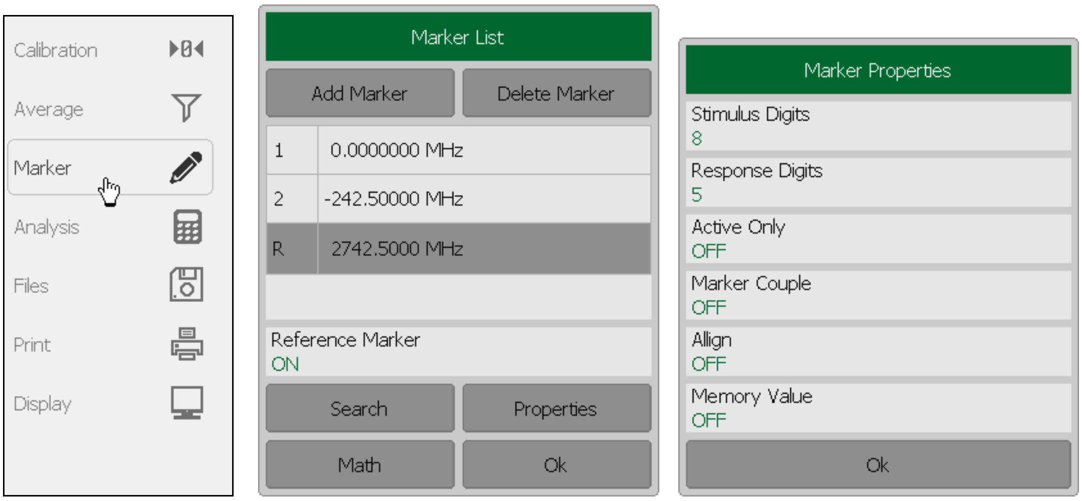
In the **Marker Properties** dialog click on the **Marker Couple** value field to toggle between the values.

Close the dialog by **Ok.**

6.1.6.2 Marker Value Indication Capacity

By default, the marker stimulus values are displayed with 8 decimal digits and marker response values are displayed with 5 decimal digits. The user can change these settings. The stimulus range is from 5 to 10 decimal digits, and response range is from 3 to 8 decimal digits.

To set the marker value indication capacity use the following softkeys **Marker > Properties.**

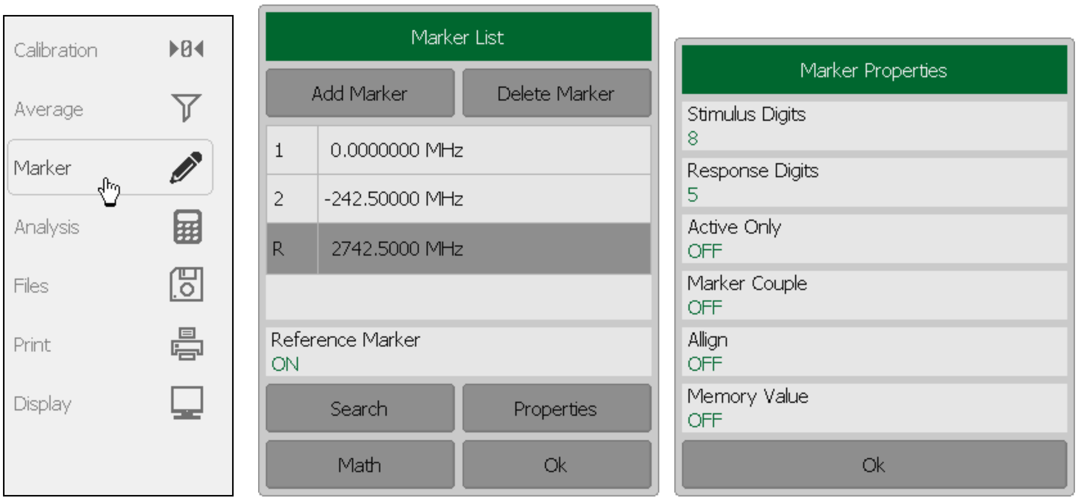


Click on the **Stimulus Digits** field to enter the number of stimulus decimal digits. Click on the **Response Digits** field to enter the number of response decimal digits. Close the dialog by **Ok.**

6.1.6.3 Multi Marker Data Display

If several traces are displayed in one channel window, by default only the active trace marker data are displayed on the screen. The user can enable displaying marker data of all traces simultaneously. The markers of different traces will be distinguished by color. Each marker will have the same color with its trace.

To enable/disable the multi marker data display use the softkeys **Marker > Properties.**



Click in the **Active Only** field.

The **OFF** value stands for multi marker data display mode.

Note	When multi marker data display is enabled, arrange the marker data on the screen to avoid data overlapping.

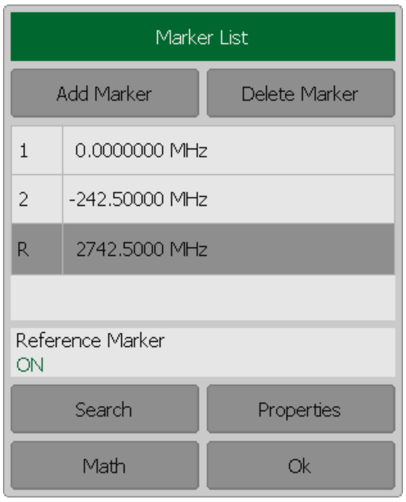
6.1.6.4 Marker Data Alignment

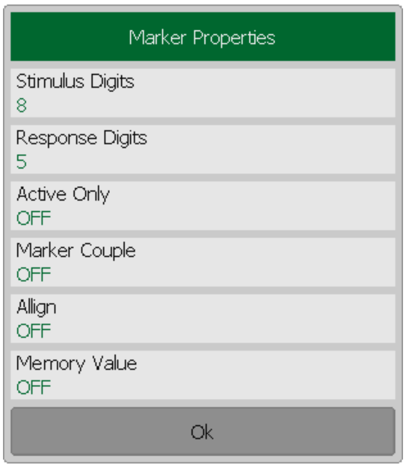
By default marker data are arranged individually for each trace. The user can enable marker data alignment on the screen. Such alignment cancels individual arrangement of marker data of different traces. The marker data of all succeeding traces are aligned against the first trace.

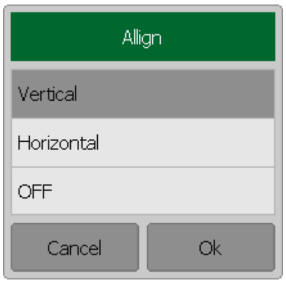
There are two types of alignment:

- Vertical – marker data of different traces are arranged one under another;
- Horizontal – marker data of different traces are arranged in a line.

To enable marker data alignment use the following softkeys **Markers > Properties.**







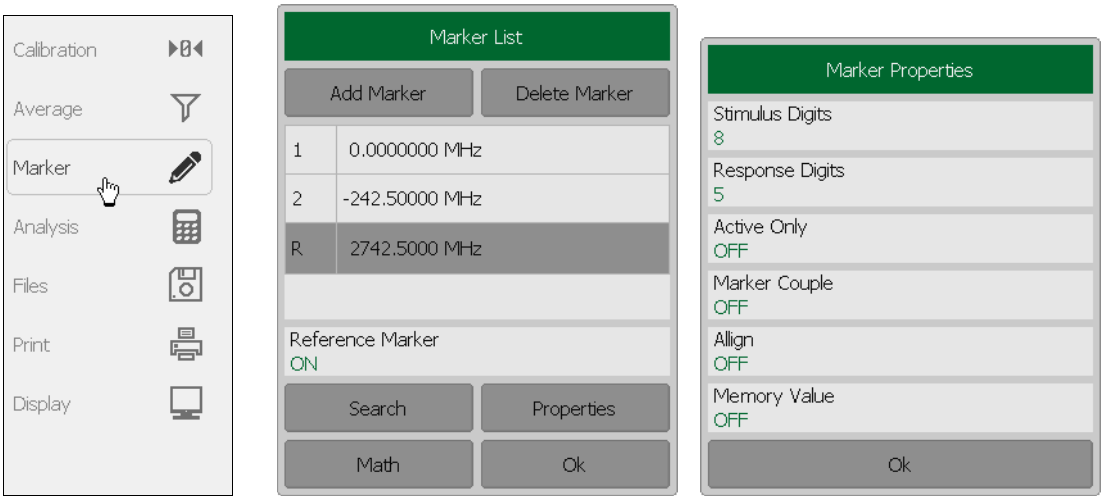
Click in the **Align** parameter value field. In the **Align** dialog, double click on the alignment type.

Close the dialog by clicking Ok.

6.1.6.5 Memory trace value display

By default the marker values of the data traces (not memory traces) are displayed on the screen. The user can enable the display of memory trace marker values, if a memory trace is available.

To enable/disable the display of memory trace marker values, toogle the following softkeys **Marker > Properties > Memory Value.**



When the display of memory trace marker values is on, the marker indicates the stored data at the same time with the current. Marker pointers appear on the memory trace are the same as on the data trace. Markers pointers are interactive. They can be moved with the mouse to watch the stored data.

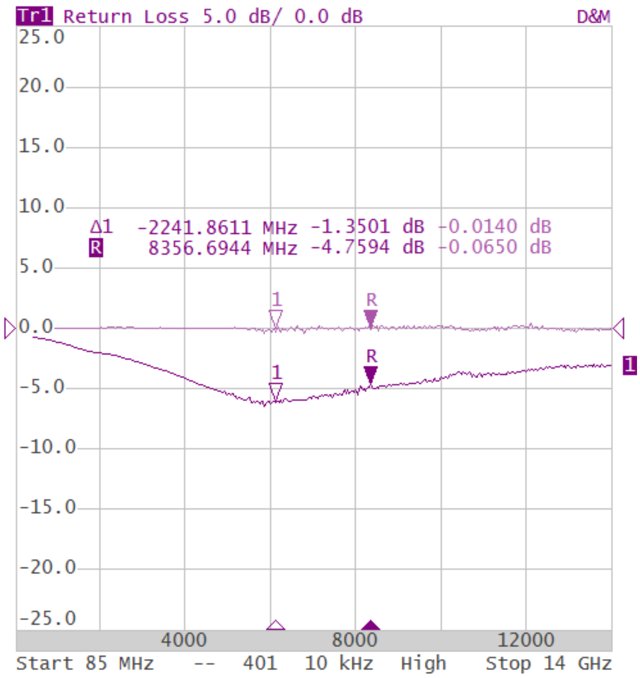


Figure 6.4 Display of the memory value using markers

6.1.7 Marker Position Search Functions

Marker position search function enables you to find the following values on a trace:

- maximum value;

- minimum value;
- peak value;
- target level.

Before you start the search, first activate the marker.

6.1.7.1 Search for Maximum and Minimum

Maximum and minimum search functions enable you to determine the maximum and minimum values of the measured parameter and move the marker to these positions on the trace (see Figure 6.5).

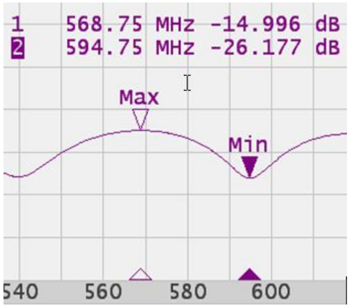
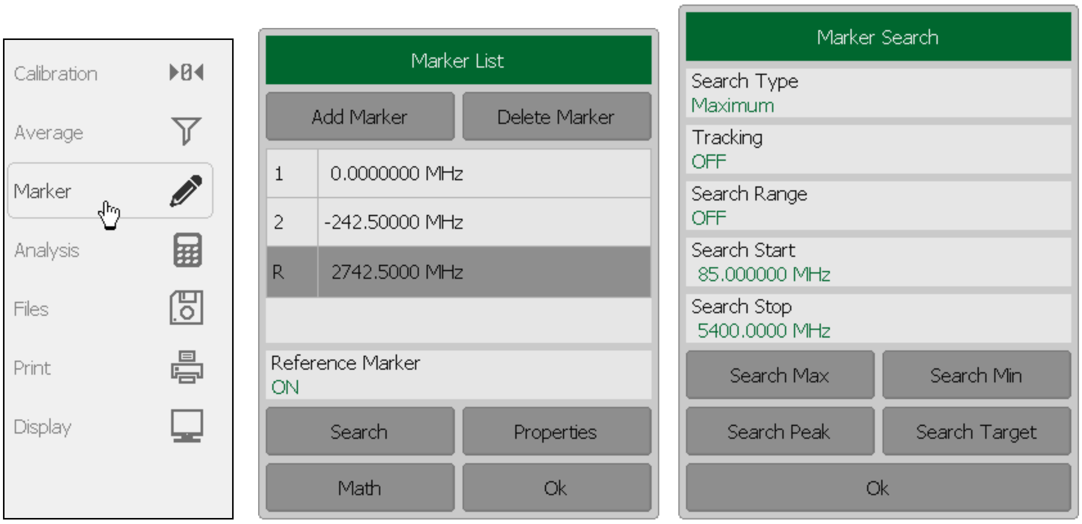


Figure 6.5 Maximum and minimum search

To find the maximum or minimum values on a trace use the following softkeys: **Marker > Search > Search Min | Search Max**



The last search type applied to the marker is indicated in the **Search Type** field of the Search dialog.

6.1.7.2 Search for Peak

Peak search function enables you to determine the peak value of the measured parameter and move the marker to this position on the trace (see Figure 6.6).

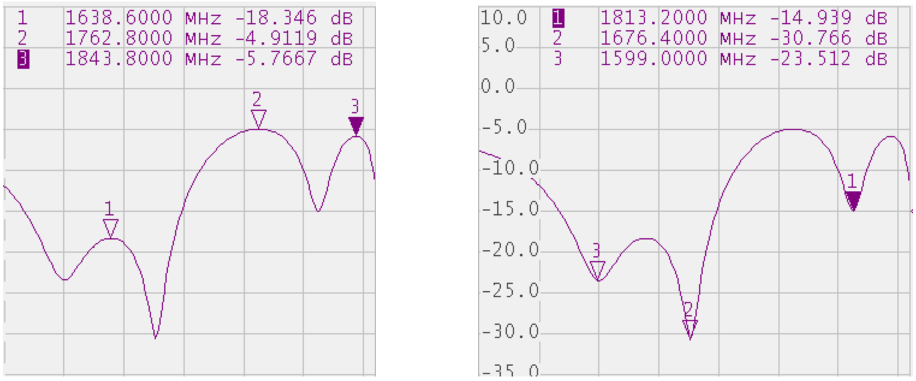


Figure 6.6 Positive and negative peaks

Peak is called **positive** if the value in the peak is greater than the values of the adjacent points.

Peak is called **negative** if the value in the peak is smaller than the values of the adjacent points.

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 two following conditions:

- The peaks must have the polarity (positive, negative, or both) specified by the user;
- The peaks must have the peak deviation not less than the value assigned by the user.

The following options of the peak search are available:

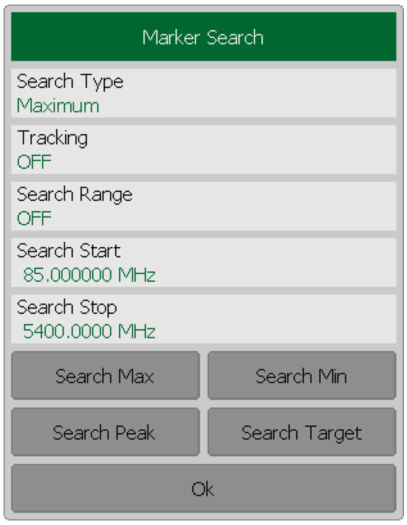
- Search for the nearest peak;
- Search for the greatest peak;
- Search for the left peak;
- Search for the right peak.

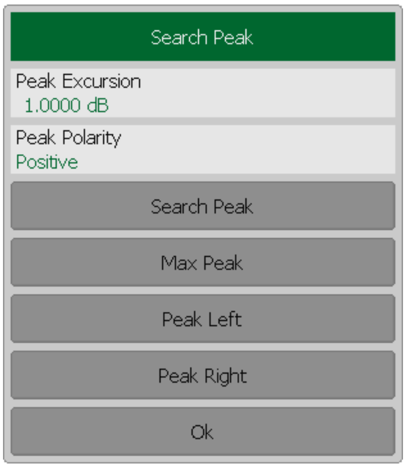
The nearest peak is the peak, which is located nearest to the current position of the marker along the stimulus axis.

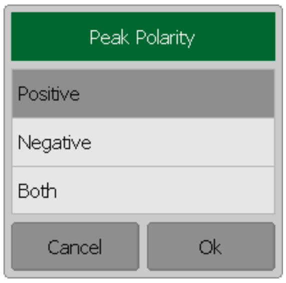
The greatest peak is a peak with maximum or minimum value, depending on the current polarity settings of the peak.

Note	The search for the greatest peak is different from the search for maximum or minimum as the peak cannot be located in the limiting points of the trace even if these points have maximum or minimum values.

To search for the peak value use the following softkeys **Marker > Search > Search Peak.**







Depending on the search function select one of the following softkeys:

- **Search Peak;**
- **Max Peak;**
- **Peak Left;**
- **Peak Right.**

Set the peak excursion value if necessary. Click on the **Peak Excursion** field and set the required peak polarity by a click in the **Peak Polarity** field.

6.1.7.3 Search for Target Level

Target level search function enables you to locate the marker with the given (target) level of the measured parameter (see Figure 6.7).

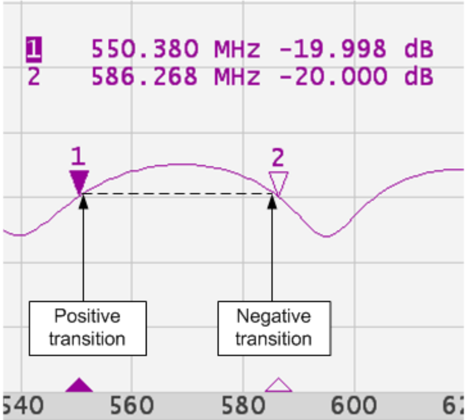


Figure 6.7 Target level search

The trace can have two types of transition in the points where the target level crosses the trace:

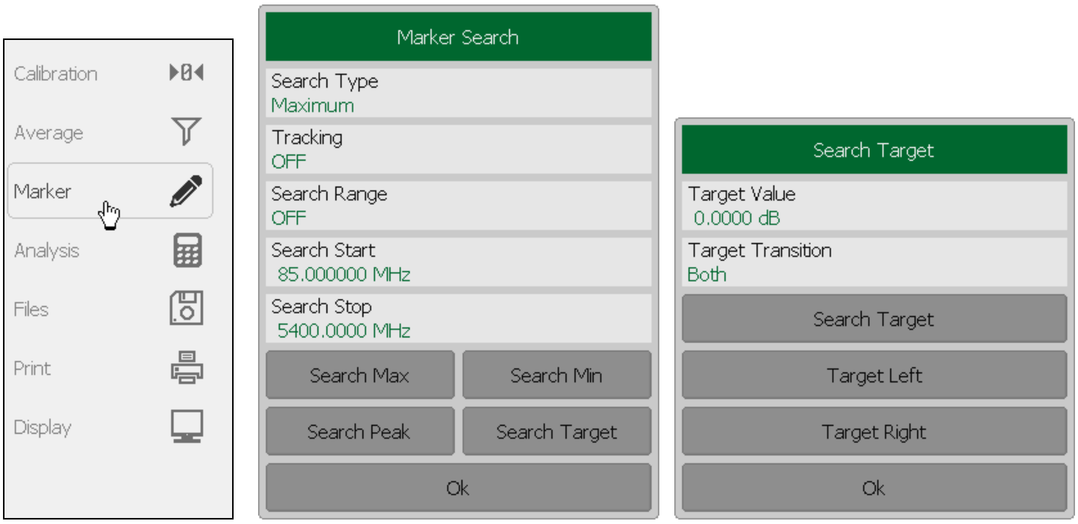
- transition type is positive if the function derivative (trace slope) is positive at the intersection point with the target level;
- transition type is negative if the function derivative (trace slope) is negative at the intersection point with the target level.

The target level search is executed only for the intersection points, which have the specific transition polarity selected by the user (positive, negative or both).

The following options of the target level search are available:

- Search for the nearest target;
- Search for the left target;
- Search for the right target.

To search for target level value use the following softkeys **Marker > Search > Search Target.**



Depending on the search function select one of the following softkeys:

- **Search Target;**
- **Target Left;**
- **Target Right.**

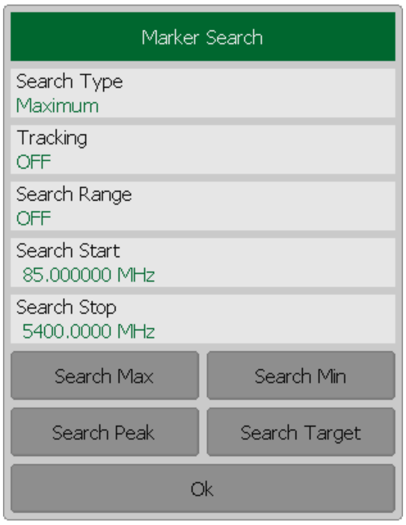
To set the target level value click on the **Target Value** field and enter the value using the on-screen keypad.

To set the transition type click on the **Target Transition** field.

6.1.7.4 Search Tracking

The marker position search function by default can be initiated by any search softkey. Search tracking mode allows you to perform continuous marker position search, until this mode is disabled.

To enable/disable search tracking mode use the following softkeys **Marker > Search.**



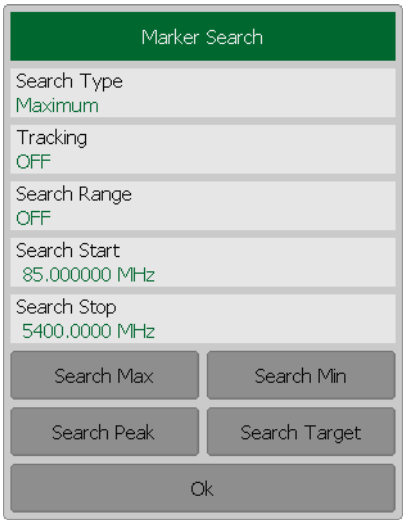
Click on the **Tracking** field to enable/disable the search tracking mode.

Tracking will be performed for the last searched marker search type. The marker search type will be indicated in the **Search Type** field.

6.1.7.5 Search Range

The user can set the search range for the marker position search by setting the stimulus limits.

To enable/disable the search range use the following softkeys **Marker > Search.**



Click on the **Search Range** field to enable/disable the search range.

To enter the search range parameters click on the **Search Start** or **Search Stop** field and enter the stimulus value using the on-screen keypad.

6.1.8 Marker Math Functions

Marker math functions are the functions, which use markers for calculating of various trace characteristics. Four marker math functions are available:

- Statistics;
- Bandwidth Search;
- Flatness;
- RF Filter.

6.1.8.1 Trace Statistics

The trace statistics feature allows the user to determine and view such trace parameters as mean, standard deviation, and peak-to-peak. The trace statistics range can be defined by two markers (see Figure 6.8).

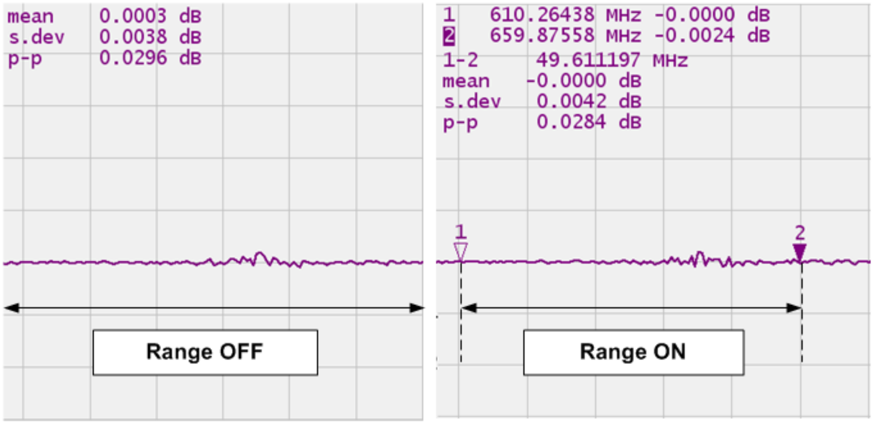
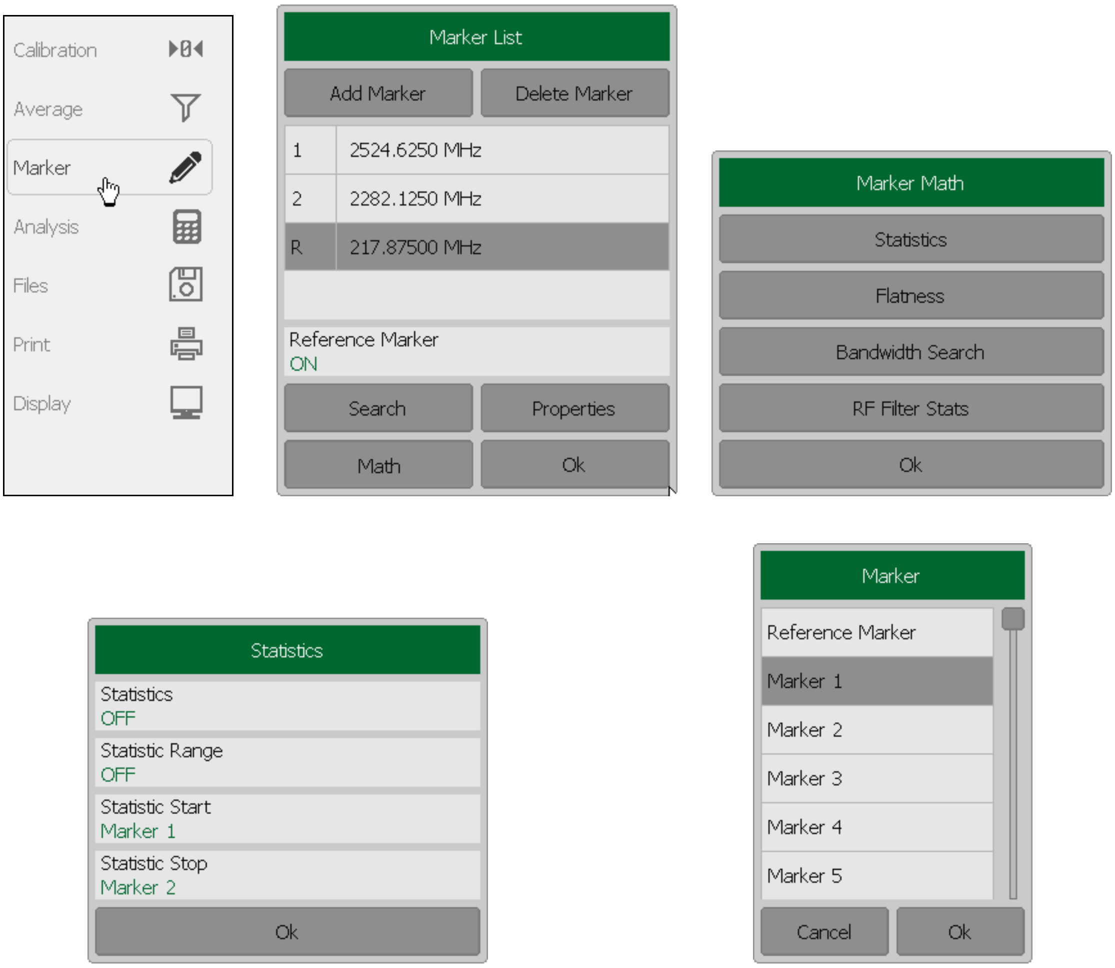


Figure 6.8 Trace statistics

Table 6.1 Statistics parameters

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 use the following softkeys **Markers > Math > Statistics.**



Click on the **Statistics** field to toggle between the on/off status.

To enable/disable statistics range feature click on the **Statistics Range** field to toggle between the on/off status.

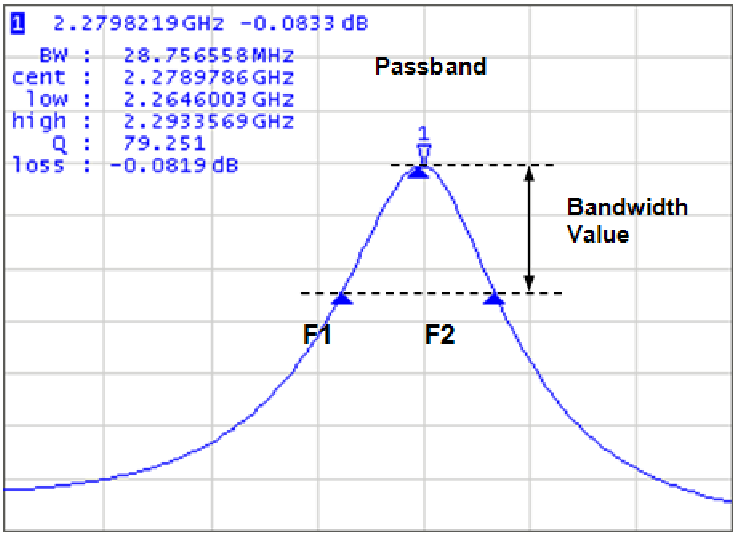
The statistics range is set by two markers. If there are no markers in the list, add two markers. Marker adding operation is described in section 6.1.1

Click on the **Statistic Start** or **Statistic Stop** field and select the required marker numbers from the list.

6.1.8.2 Bandwidth Search

The *bandwidth search* function allows the user 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 (See Figure 6.9). In the figure, F1 and F2 are the lower and higher cutoff frequencies of the band respectively.

The bandwidth search is executed from the reference point. The user can select as reference point the active marker or the maximum of the trace. The bandwidth search function determines the lower and higher cutoff frequencies, which are apart from the reference point response by *bandwidth value* defined by the user (usually −3 dB).



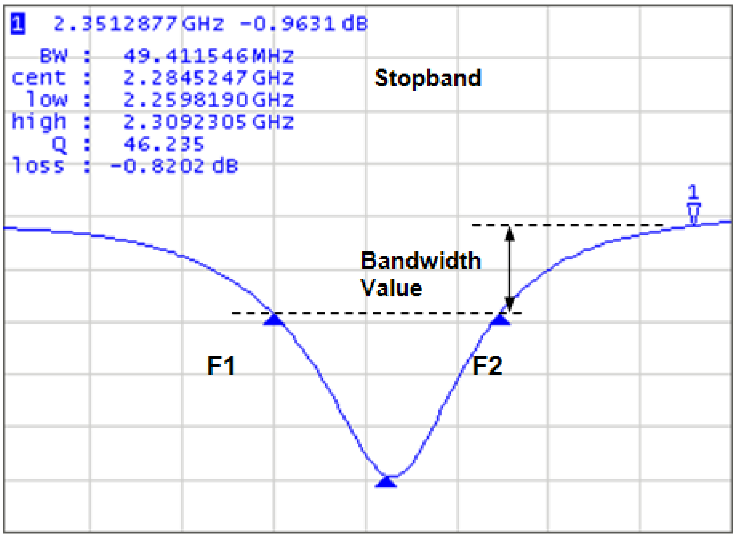
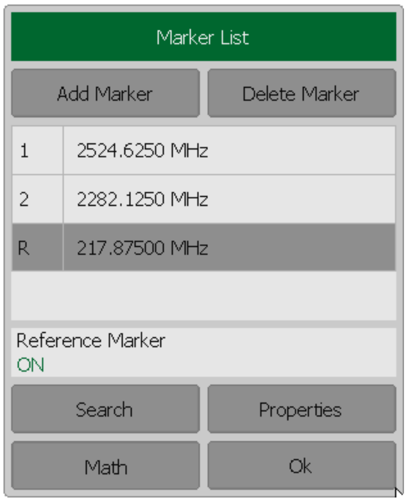


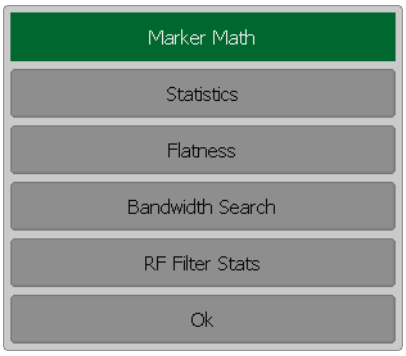
Figure 6.9 Bandwidth search

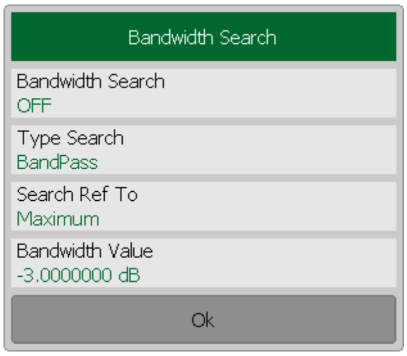
Table 6.2 Bandwidth parameters

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	-

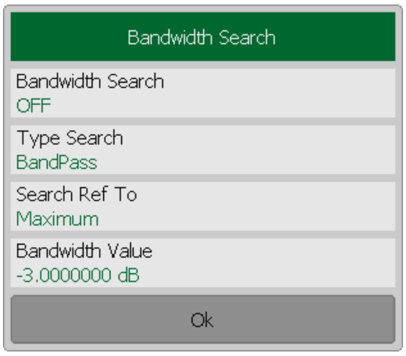
To enable/disable bandwidth search function, use the following softkeys: **Marker > Math > Bandwidth Search > Bandwidth Search.**





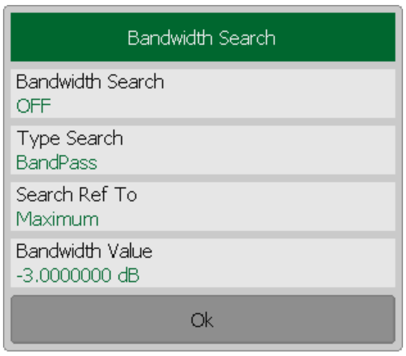


Set the bandwidth search type by softkeys: **Marker > Math > Type Search.**



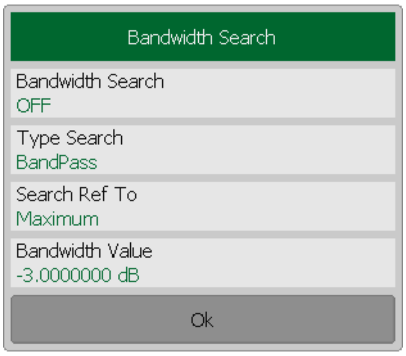
The type and the softkey label toggle between **Bandpass** and **Notch** settings.

To set the search reference point, use the following softkeys: **Marker > Math > Bandwidth Search > Search Ref To.**



The type and the softkey label toggle between **Maximum, Marker** and **Minimum** settings.

To enter the bandwidth value, use the following softkeys: **Marker > Math > Bandwidth Search > Bandwidth Value.**



6.1.8.3 Flatness

The flatness function allows the user to determine and view the following trace parameters: gain, slope, and flatness. The user sets two markers to specify the flatness search range (see Figure 6.10).

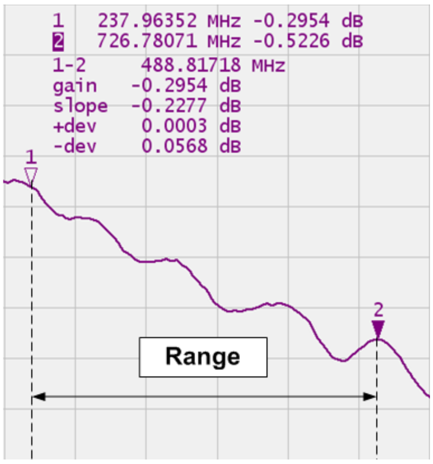


Figure 6.10 Flatness

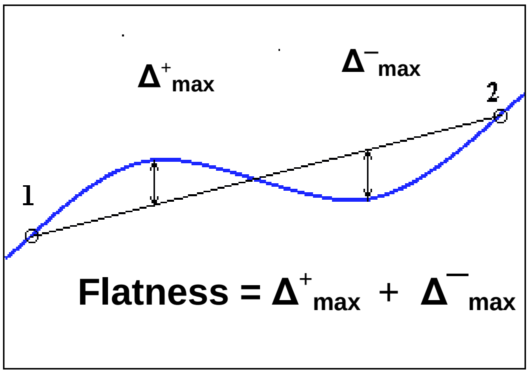
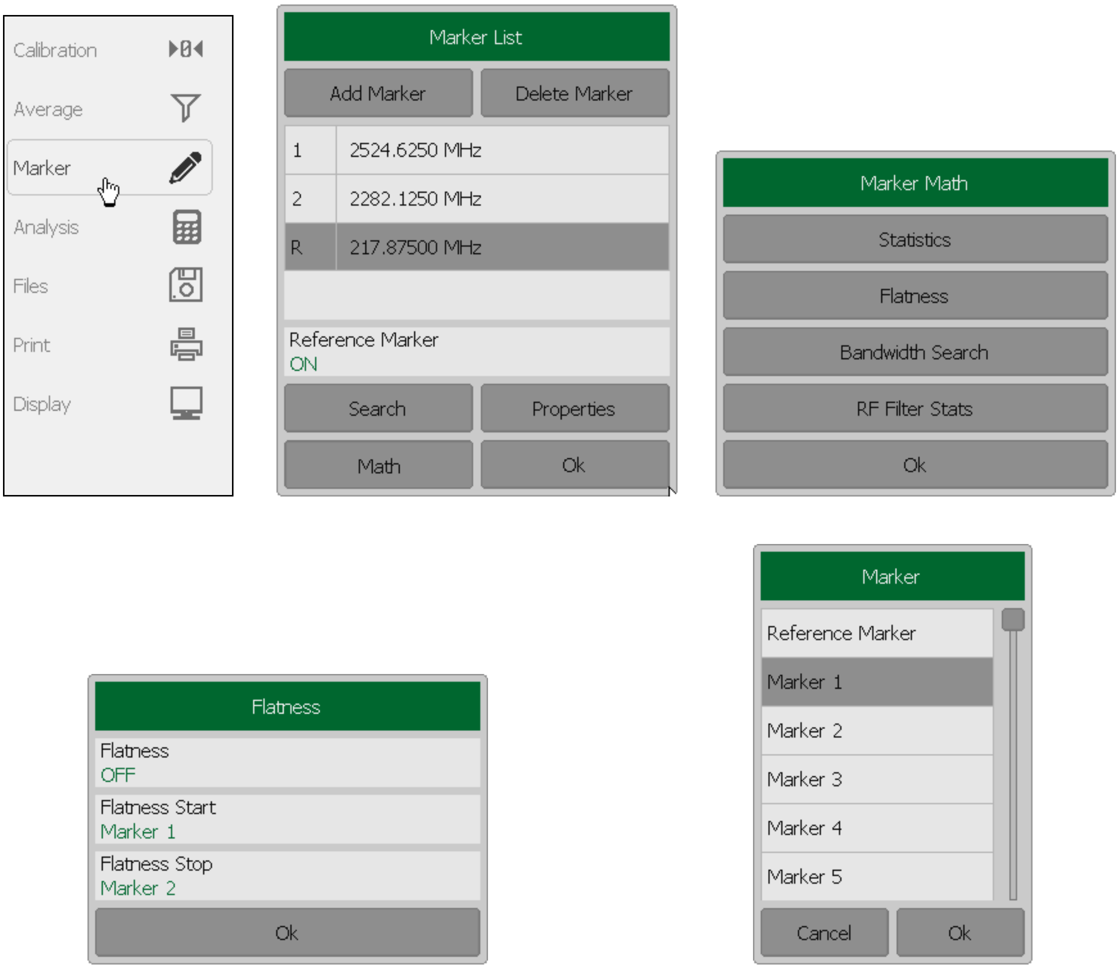


Figure 6.11 Flatness parameters determination

Table 6.3 Flatness parameters

Parameter Description	Symbol	Definition
Gain	**gain**	Marker 1 value
Slope	**slope**	Difference between marker 2 and marker 1 values.
Flatness	**+dev** **-dev**	Sum of “positive” and “negative” peaks of the trace, which are measured from the line connecting marker 1 and marker 2 (see Figure 6.11).

To enable/disable the flatness search function use the following softkeys **Markers > Math > Flatness.**



Click on the **Flatness** field to toggle between the on/off status.

Flatness range is set by two markers. Add two markers, if there are no markers in the list. Marker adding procedure is described in section 6.1.1.

Click on the **Flatness Start** or **Flatness Stop** field and select the required marker numbers from the list.

6.1.8.4 RF Filter Statistics

The *RF filter statistics* function allows the user 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, the stopband is specified by the second pair of markers (See Figure. 6.12).

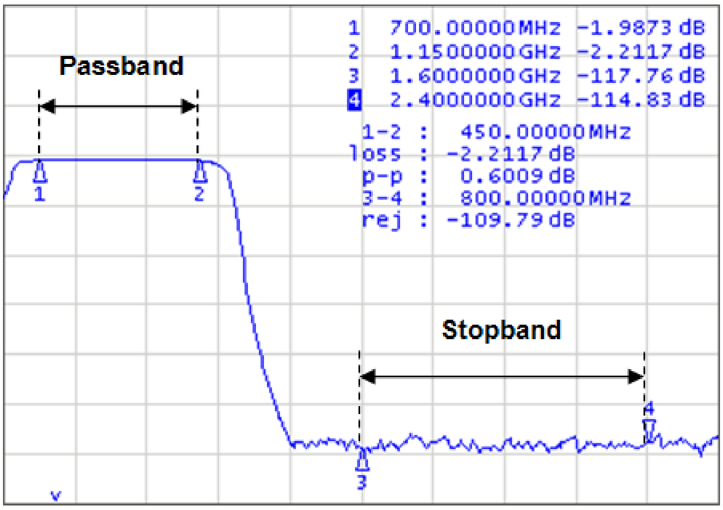
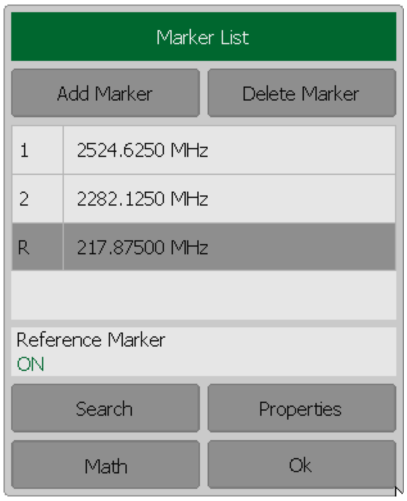


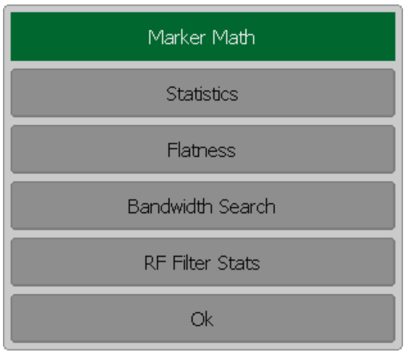
Figure. 6.12 RF filter statistics

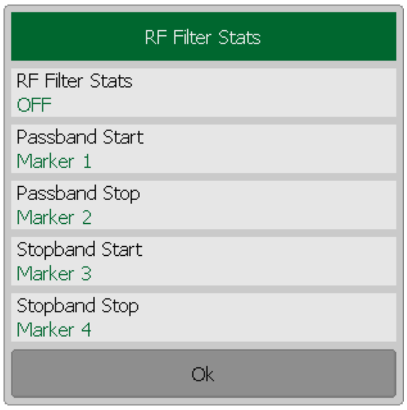
Table 6.4 RF filter statistics parameters

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

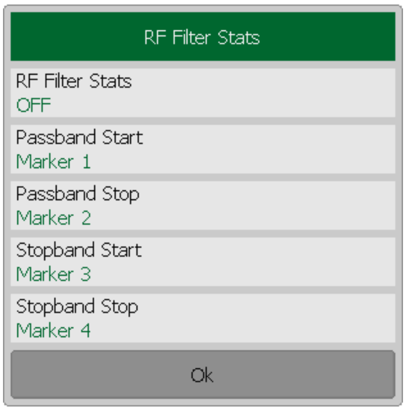
To enable/disable the RF filter statistics function, use the following softkeys: **Marker > Math > RF Filter Stats > RF Filter Stats.**

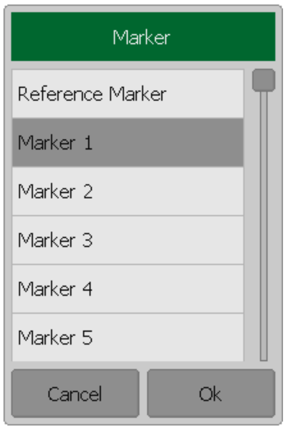




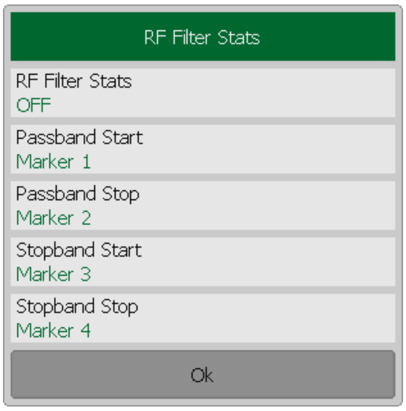


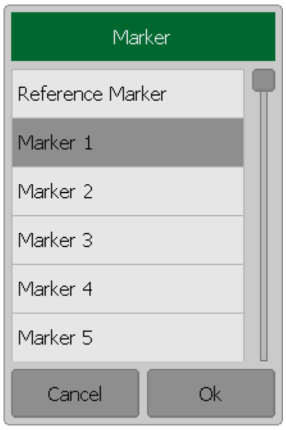
To select the markers specifying the passband, use the following softkeys: **Marker > Math > RF Filter Stats > Passband Start | Passband Stop.**





To select the markers specifying the stopband, use the following softkeys: **Marker > Math > RF Filter Stats > Stopband Start | Stopband Stop.**





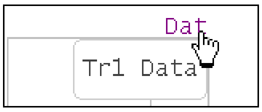
6.2 Memory Trace Function

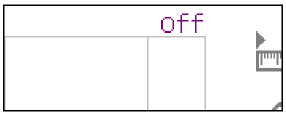
For each data trace displayed on the screen a so-called memory trace can be created. Memory traces can be saved for each data trace. The memory trace is displayed in the same color as the main data trace, but its brightness is lower.

The memory trace is a data trace saved into the memory. It is created from the current measurement when the user is clicking the corresponding softkey or when the current sweep is completed. After that, the two traces become simultaneously displayed on the screen – the data trace and the memory trace.

The trace status field will indicate the following:

- **Dat** – only data trace is displayed;
- **Mem** – only memory trace is displayed;
- **D&M** – data trace and memory trace are displayed;
- **Off** - both traces are not displayed





The memory trace has the following features of the data trace:

- frequency range,
- number of points,
- sweep type.

The memory trace has the following settings common with the data trace (which if changed, modifies both traces):

- format,
- scale,
- smoothing,
- electrical delay.

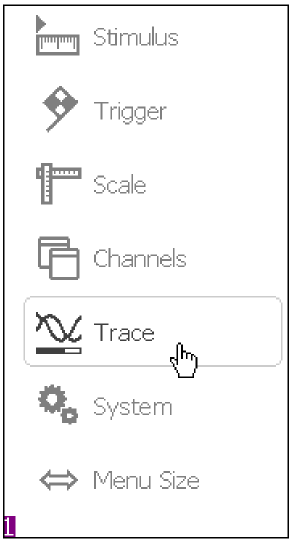
The following data trace settings (if changed after the memory trace creation) do not influence the memory trace:

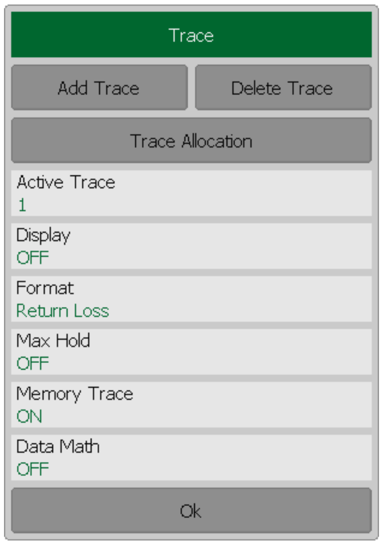
- power in frequency sweep mode,
- IF bandwidth,
- averaging,
- calibration.

6.2.1 Saving Trace into Memory

The memory trace function can be applied to the individual traces of the channel. Before you enable this function, first activate the trace.

Click the following softkey in the left-hand menu bar **Trace**





The active trace will be highlighted in the list. If necessary select the required trace by clicking on it.

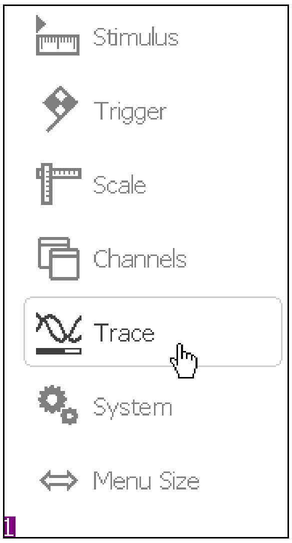
To enable trace saving into memory click on the **Memory Trace** field to set the value to **ON**.

The data will be saved into memory immediately.

6.2.2 Memory Trace Deleting

The memory trace deleting can be applied to the individual traces of the channel. Before you enable this function you have to activate the trace.

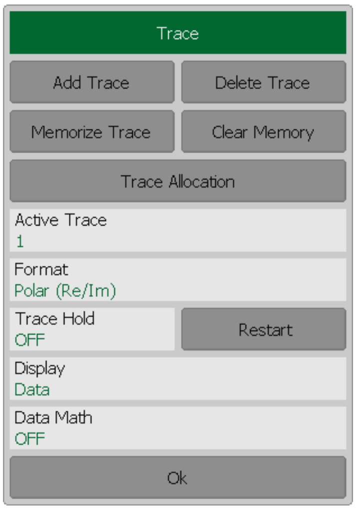
Click the **Trace** softkey in the right menu bar

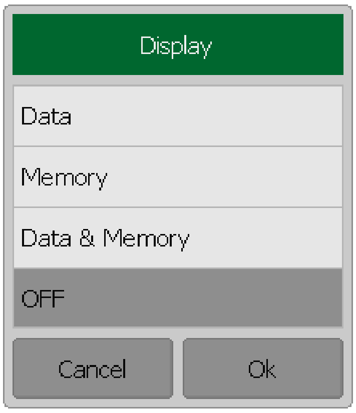


To delete a memory trace click in the **Memory Trace** parameter value field. The **Memory Trace** parameter value will change to **OFF**.

6.2.3 Trace Display Setting

To set the type of data to be displayed on the screen, use the following softkeys **Trace > Display > Data | Memory | Data & Memory | OFF.**





Close the dialog by **Ok.**

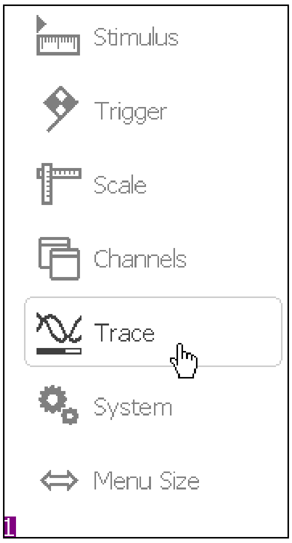
6.2.4 Memory Trace Math

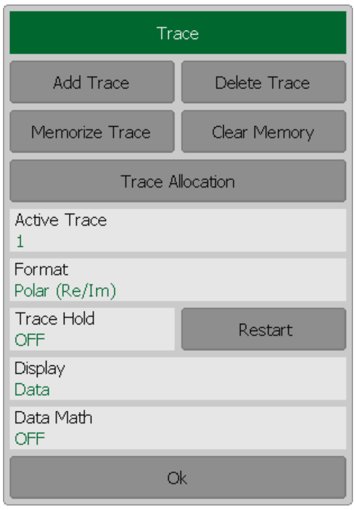
The memory trace can be used for math operations with the data trace. The resulting trace of such an operation will replace the data trace. The math operations with memory and data traces are performed in complex values. The following four math operations are available:

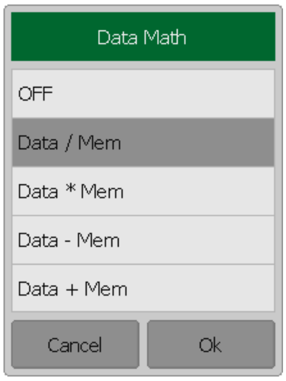
- Division of data trace by memory trace. Trace status bar indicates : **D/M**.
- Multiplication of data trace by memory trace. Trace status bar indicates: **D*M**.
- Subtraction of memory trace from data trace. Trace status bar indicates: **D–M**.
- Addition of data trace and memory trace. Trace status bar indicates: **D+M**.

The memory trace function can be applied to individual traces of the channel. Before you enable this function, first activate the trace.

Click the following softkey in the right menu bar **Trace.**







Click the **Data Math** field. In the **Data Math** dialog select the math operation type for the current data traces and memory traces. Close the dialog by **Ok.**

The result of math operation will be displayed in the form of current data traces.

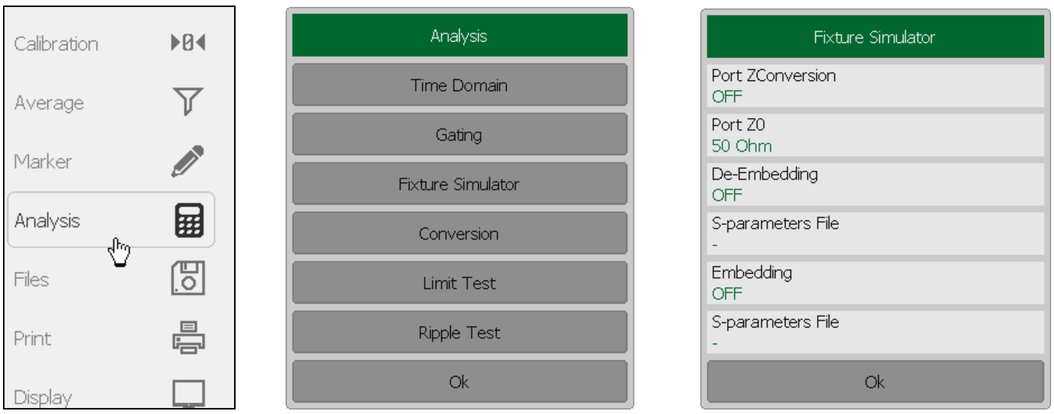
6.3 Fixture Simulation

The fixture simulation function enables you to emulate the measurement conditions other than those of the real setup. The following conditions can be simulated:

- Port Z conversion;
- De-embedding;
- Embedding.

Before starting the fixture simulation, first activate the channel. The simulation function will affect all the traces of the channel.

To open the fixture simulation menu use the following softkeys **Analysis > Fixture Simulator.**

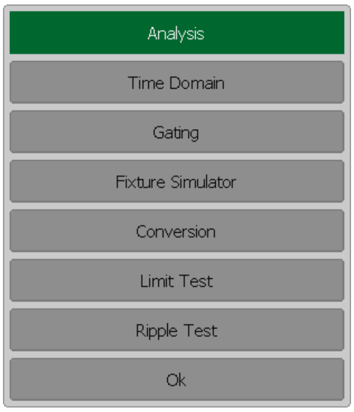


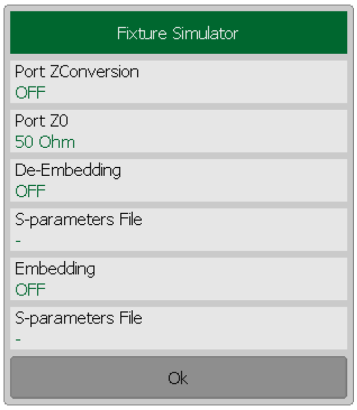
6.3.1 Port Z Conversion

Port Z conversion is a function of transformation of the S-parameters measured during port wave impedance change simulation.

Note	The value of the test port impedance is defined in the process of calibration. It is determined by the characteristic impedance of the calibration kit.

To open the fixture simulation menu use the following softkeys **Analysis > Fixture Simulator.**





To enable/disable the port impedance conversion function click on the **Port Z Conversion** field.

To enter the value of the simulated impedance of Port click on the **Port Z0** field and enter the value using the on-screen keypad.

6.3.2 De-embedding

De-embedding is a function of the S-parameter transformation by removing of some circuit effect from the measurement results.

The circuit being removed should be defined in the data file containing S-parameters of this circuit. The circuit should be described as a 2-port in Touchstone file (extension .s2p), which contains the S-parameter table: S11, S21, S12, S22 for a number of frequencies.

The de-embedding function allows to exclude mathematically 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 of the network with this circuit removed (see Figure 6.13).

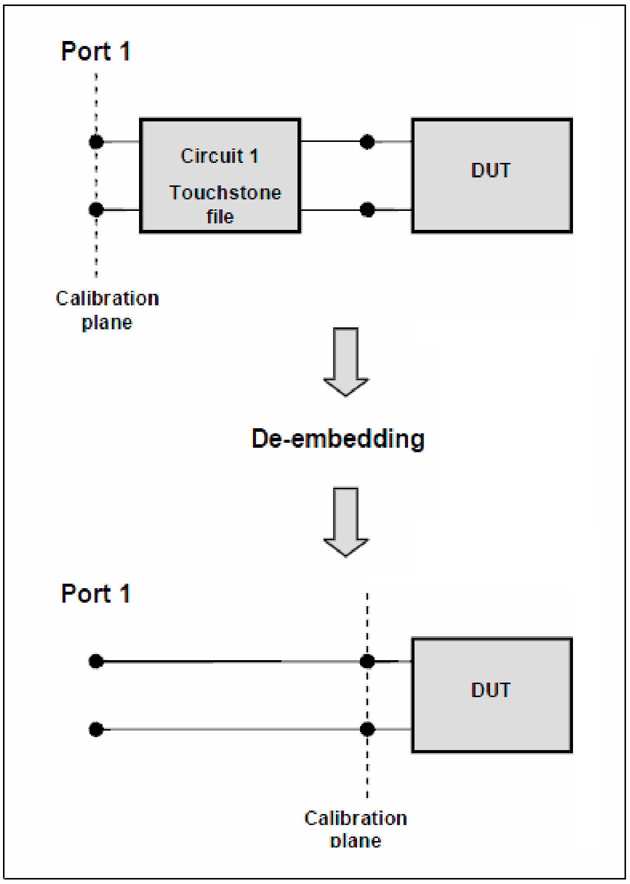
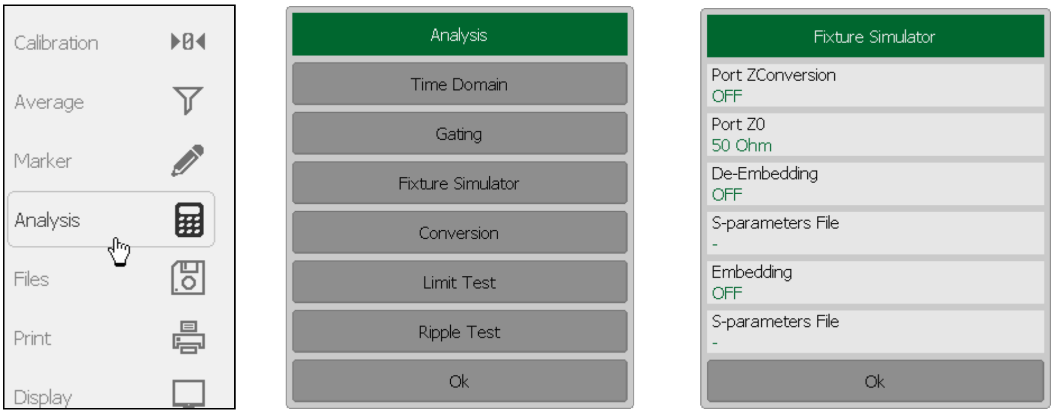


Figure 6.13 De-embedding

To enable/disable the de-embedding function for port 1 use the following softkeys **Analysis > Fixture Simulator.**



And click on the **De-Embedding** field to toggle between the on/off status. To enter the file name of the de-embedded circuit S – parameters of port 1 click on the **S - parameters File** field.

Note	If S-parameters file is not specified, the field of the function activation will be grayed out.

6.3.3 Embedding

Embedding is a function of the S-parameter transformation by integration of some virtual circuit into the real network (see Figure 6.14). The embedding function is an inverted de-embedding function.

The circuit being integrated should be defined in the data file containing S-parameters of this circuit. The circuit should be described as a 2-port in Touchstone file (extension .s2p), which contains the S-parameter table: S11, S21, S12, S22 for a number of frequencies.

The embedding function allows to simulate mathematically the DUT parameters after adding of the fixture circuits.

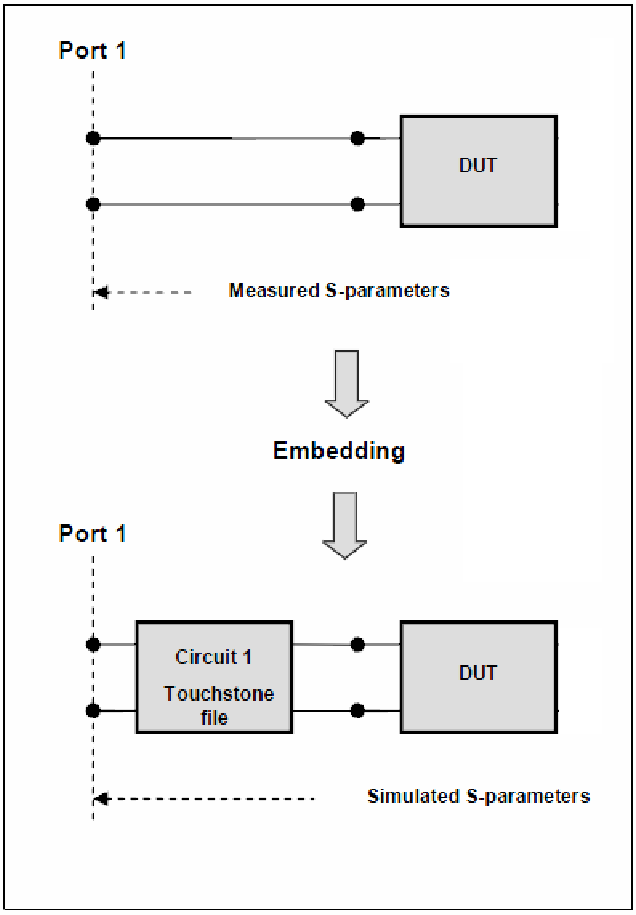
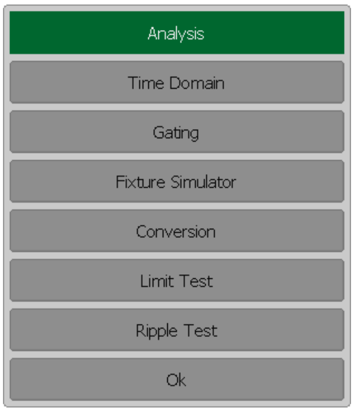
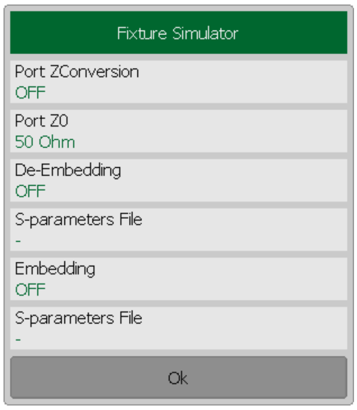


Figure 6.14 Embedding

To enable/disable the embedding function for port 1 use the following softkeys **Analysis > Fixture Simulator.**





And click on the **Embedding** field to toggle between the on/off status.

To enter the file name of the embedded circuit S – parameters of port 1 click on the **S - parameters File** field.

Note	If S-parameters file is not specified, the field of the function activation will be grayed out.

6.4 Time Domain Transformation

The Analyzer measures and displays parameters of the DUT in frequency domain. Time domain transformation is a function of mathematical modification of the measured parameters in order to obtain the time domain representation.

For time domain transformation Z-transformation and frequency domain window function are applied.

The time domain transformation can be activated for separate traces of a channel. The current frequency parameters S_{11} of the trace will be transformed into the time domain.

Note	Traces in frequency and time domains can simultaneously belong to one channel. The stimulus axis label will be displayed for the active trace, in frequency or time units.

The transformation function allows setting of the measurement range in time domain within Z-transformation ambiguity range. The ambiguity range is determined by the measurement step in the frequency domain:

$$\Delta T = \frac{1}{\Delta F}; \quad \Delta F = \frac{F\max - F\min}{N-1}$$

The time domain function allows to select the following transformation types:

- Bandpass mode simulates the impulse bandpass response. It allows the user to obtain the response for circuits incapable of direct current passing. The frequency range is arbitrary in this mode. The time domain resolution in this mode is twice lower than it is in the lowpass mode;
- Lowpass mode simulates lowpass impulse and lowpass step responses. It is applied to the circuits passing direct current, and the direct component (in point F=0 Hz) is interpolated from the start frequency (Fmin) of the range. In this mode the frequency range represents a harmonic grid where the frequency value at each frequency point is an integer multiple of the start frequency of the range Fmin. The time domain resolution is twice higher than it is in the bandpass mode.

The time domain transformation function applies Kaiser window for initial data processing in frequency domain. The window function allows to reduce the ringing (side lobes) in the time domain. The ringing is caused by the abrupt change of the data at the limits of the frequency domain. But while side lobes are reduced, the main pulse or front edge of the lowpass step becomes wider.

The Kaiser window is described by β parameter, which smoothly fine-tune the window shape from minimum (rectangular) to maximum. The user can fine-tune the window shape or select one of the three preprogrammed windows:

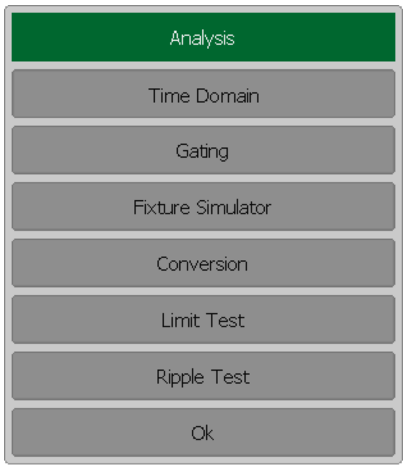
- **Minimum** (rectangular);
- **Normal**;
- **Maximum**.

Table 6.5 Preprogrammed window types

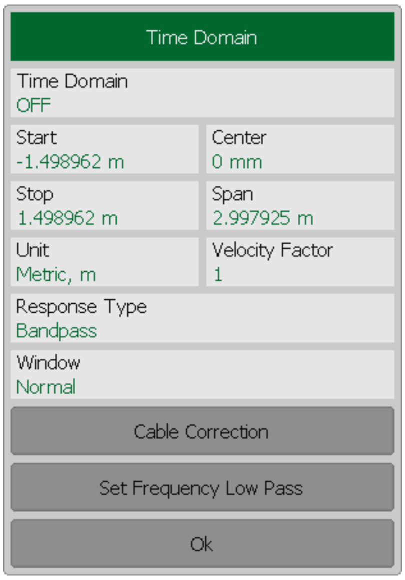
Window	Lowpass Impulse		Lowpass Step	
	Sidelobe Level	Width of the impulse (50%)[1]	Sidelobe Level	Rise time (10-90%)
Minimum	– 13 dB	$\frac{0.6}{Fmax - Fmin}$	– 21 dB	$\frac{0.45}{Fmax - Fmin}$
Normal	– 44 dB	$\frac{0.98}{Fmax - Fmin}$	– 60 dB	$\frac{0.99}{Fmax - Fmin}$
Maximum	– 75 dB	$\frac{1.39}{Fmax - Fmin}$	– 70 dB	$\frac{1.48}{Fmax - Fmin}$

6.4.1 Time Domain Transformation Activating

To enable/disable time domain transformation function, use the following softkeys: **Analysis > Time Domain > Time Domain.**



[1] The value in the band pass mode is 2 times the value in the low pass mode.

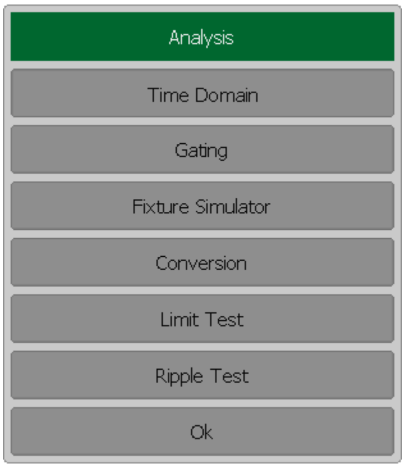


Note	Time domain transformation function is accessible only in linear frequency sweep mode.

6.4.2 Time Domain Transformation Span

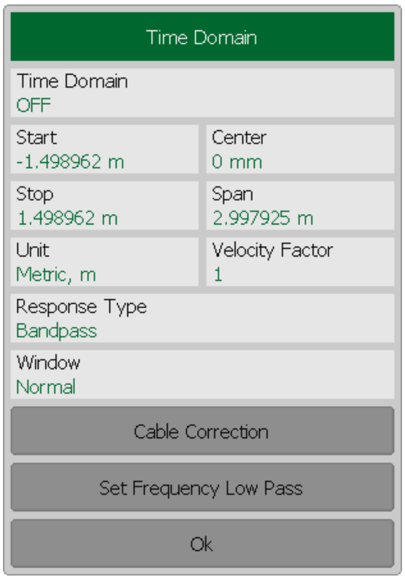
To define the span of time domain representation, you can set start and stop values.

To set the start and stop limits of the time domain range use **Analysis > Time Domain** softkeys.



Click on the **Start** or **Stop** field and enter the value using the on-screen keypad.

To set the center and span of the time domain, use the following softkeys: **Analysis > Time Domain > Center | Span.**



If velocity factor of the measured trace is known, for example in coaxial cable, the time intervals are recalculated into distances.

The transformation function allows setting of the measurement range in time domain within the limits of ambiguity range. The ambiguity range is determined by the measurement step in the frequency domain:

$$\Delta T = \frac{1}{\Delta F} = \frac{N - 1}{F_{max} - F_{min}}$$, where:

N – number of measurement points,

F_{min} – stimulus start frequency,

F_{max} – stimulus stop frequency.

The ambiguity range is recalculated into the maximum operating Distance to Fault value:

$$DTF_{max} = \frac{C \cdot V_p \cdot \Delta T}{2} = \frac{C \cdot V_p \cdot (N - 1)}{2 \cdot (F_{max} - F_{min})}$$, where:

C – velocity of light in vacuum;

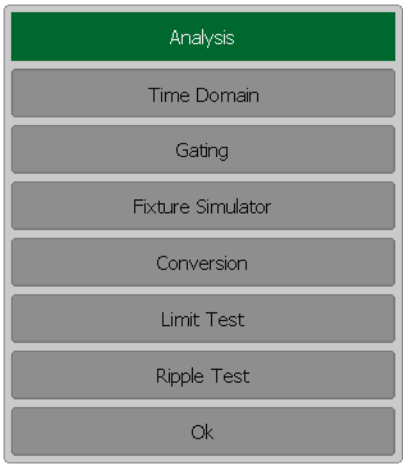
V_p – cable velocity factor.

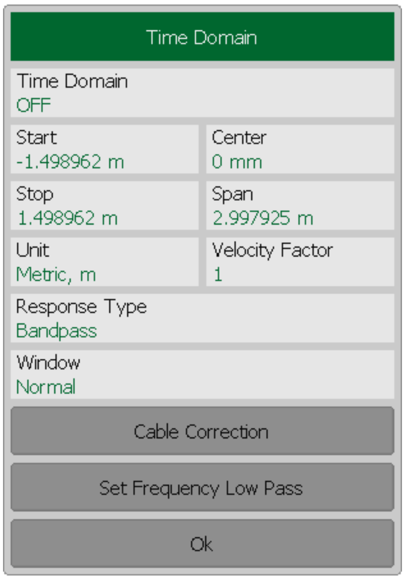
The DTF maximum value can be increased by decreasing the frequency step.

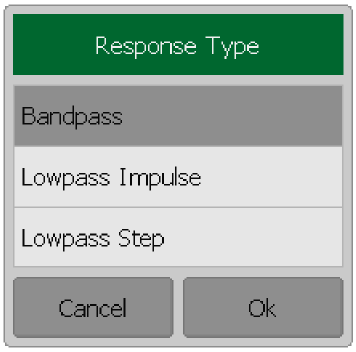
Example	If Start Freq. is 300 MHz, Stop Freq. is 600 MHz, the number of points is 10001, and velocity factor is 1, then maximum distance to fault equals is to 4996.5 m, i.e. approximately 5 km.

6.4.3 Time Domain Transformation Type

To set the time domain transformation type, use the following softkeys: **Analysis > Time Domain > Response Type > Bandpass | Lowpass Impulse | Lowpass Step.**

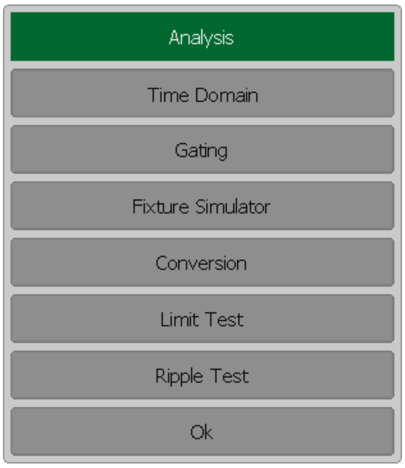


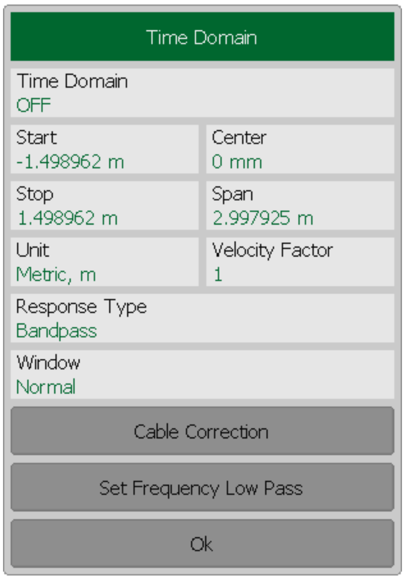




6.4.4 Time Domain Transformation Window Shape Setting

To set the window shape use the **Analysis > Time Domain** softkeys.





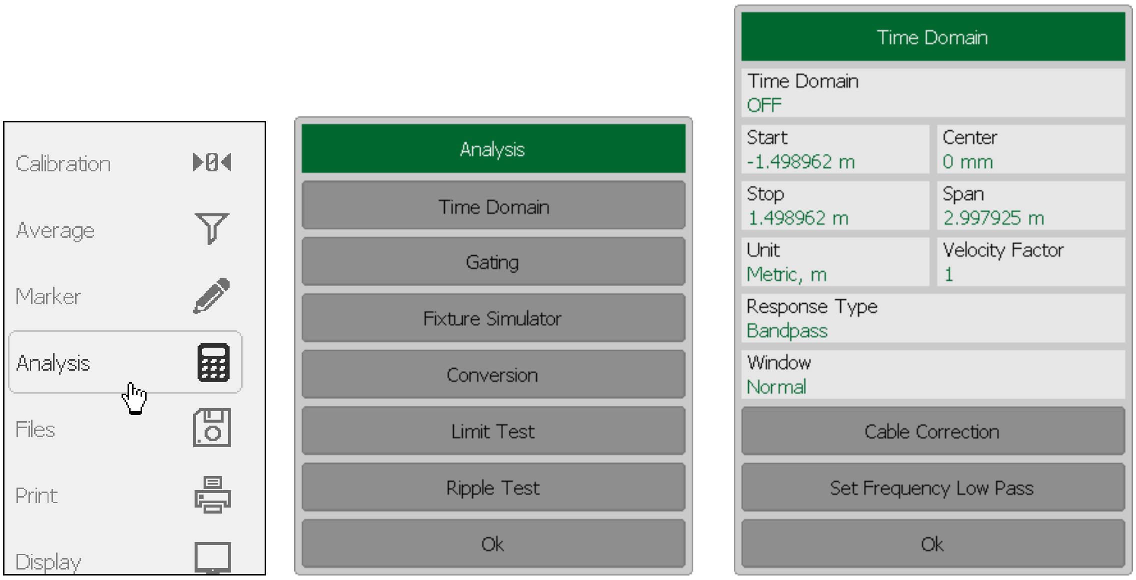
Click on the **Window** field.

Then select the required shape from the **Kaiser Window** list and complete the setting by **Ok.**

6.4.5 Frequency Harmonic Grid Setting

If lowpass impulse or lowpass step transformation is enabled, the frequency range will be represented as a harmonic grid. The frequency values in measurement points are integer multiples of the start frequency Fmin. The Analyzer is capable of creating a harmonic grid for the current frequency range automatically.

To create a harmonic grid for the current frequency range, use the following softkeys: **Analysis > Time Domain > Set Frequency Low Pass.**



Note

The frequency range will be transformed as follows:

Fmax > N x 0.02 MHz	Fmax < N x 0.02 MHz
Fmin = Fmax / N	Fmin = 0.02 MHz, Fmax = N x 0.02 MHz

6.5 Time Domain Gating

Time domain gating is a function, which mathematically removes the unwanted responses in time domain. The function performs time domain transformation and applies reverse transformation back to frequency domain to the user-defined span in time domain. The function allows the user to remove spurious effects of the fixture devices from the frequency response, if the useful signal and spurious signal are separable in time domain.

Note	Use time domain function for viewing the layout of useful and spurious responses. Then enable time domain gating and set the gate span to remove as much of spurious response as possible. After that disable the time domain function and view the response without spurious effects in frequency domain.

The function involves two types of time domain gating:

- **bandpass** – removes the response outside the gate span;
- **notch** – removes the response inside the gate span.

The rectangular window shape in frequency domain leads to spurious sidelobes due to sharp signal changes at the limits of the window. The following gate shapes are offered to reduce the sidelobes:

- **maximum;**
- **wide;**
- **normal;**
- **minimum.**

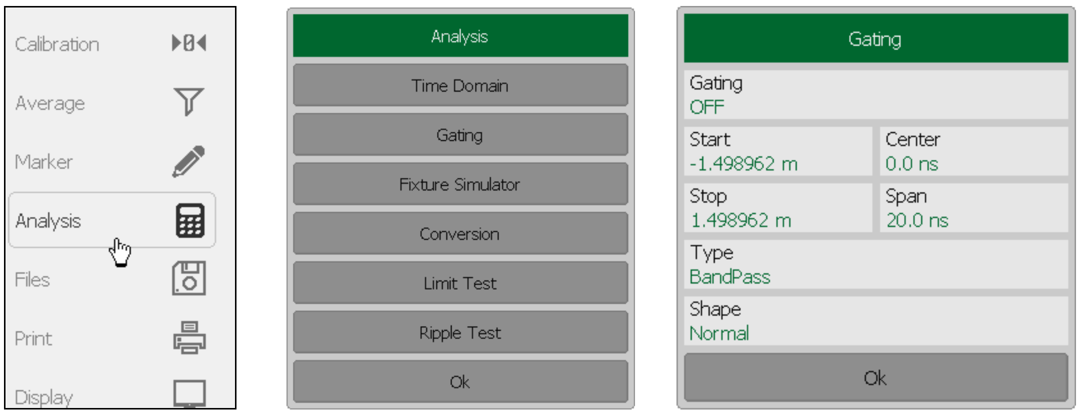
The minimum window has the shape close to rectangular. The maximum window has more smoothed shape. From minimum to maximum window shape, the sidelobe level increases and the gate resolution reduces. The choice of the window shape is always a trade-off between the gate resolution and the level of spurious sidelobes. The parameters of different window shapes are represented in Table 6.6

Table 6.6 Time domain gating window shapes

Window Shape	Bandpass Sidelobe Level	Gate Resolution (Minimum Gate Span)
Minimum	– 48 dB	$\frac{2.8}{F\text{max} - F\text{min}}$
Normal	– 68 dB	$\frac{5.6}{F\text{max} - F\text{min}}$
Wide	– 57 dB	$\frac{8.8}{F\text{max} - F\text{min}}$
Maximum	– 70 dB	$\frac{25.4}{F\text{max} - F\text{min}}$

6.5.1 Time Domain Gate Activating

To enable/disable the time domain gating function: toggle the following softkey **Analysis > Gating.**



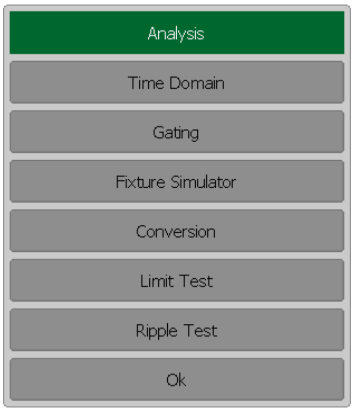
Click on the **Gating** field to toggle between the on/off settings.

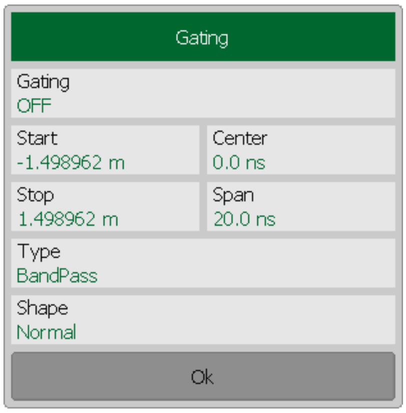
Note	Time domain gating function is accessible only in linear frequency sweep mode.

6.5.2 Time Domain Gate Span

To define the span of time domain gate, you can set its start and stop values.

To set the start and stop of the time domain gate use the following softkeys **Analysis > Gating.**



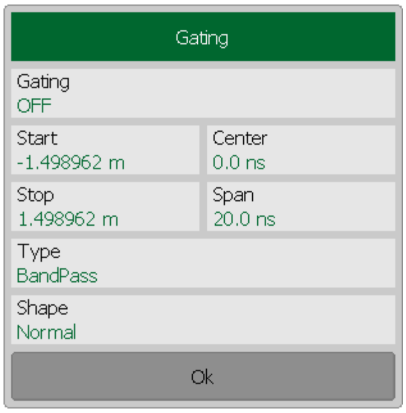


Click on the **Start** or **Stop** field and enter the value using the on-screen keypad

To set the center and span of the time domain gate, use the following softkeys: **Analysis > Gating > Center | Span.**

6.5.3 Time Domain Gate Type

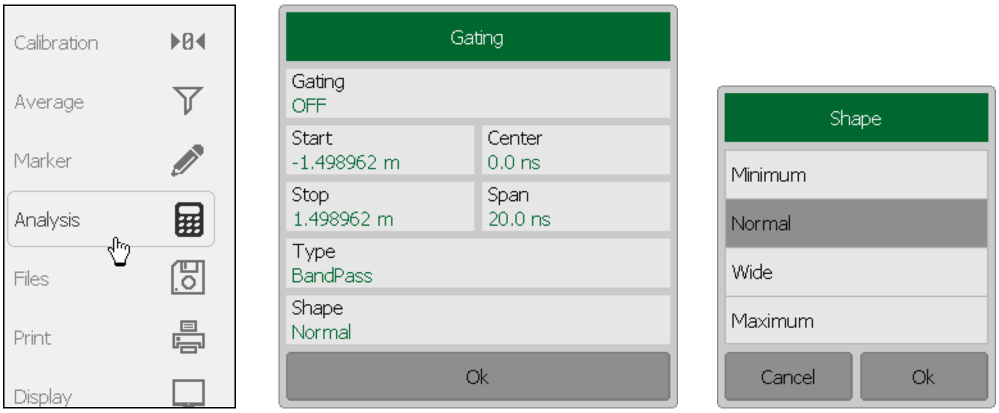
To select the type of the time domain window use the following softkeys: **Analysis > Gating.**



Click on the **Type** field to toggle the type between **Bandpass** and **Notch**.

6.5.4 Time Domain Gate Shape Setting

To set the time domain gate shape use the following softkeys **Analysis > Gating.**



Click on the **Shape** field to select the shape between **Minimum, Normal, Wide** or **Maximum.**

6.6 S-Parameter Conversion

S-parameter conversion function allows conversion of the measurement results (S_{11}) to the following parameters:

Equivalent impedance (Zr) and equivalent admittance (Yr) in reflection measurement:

$$Z_r = Z_0 \cdot \frac{1+S_{11}}{1-S_{11}}$$

$$Y_r = \frac{1}{Z_r}$$

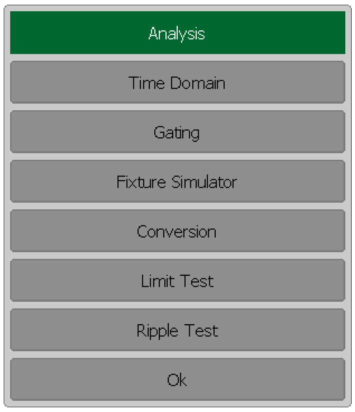
Inverse S-parameter (1/S) for reflection measurements:

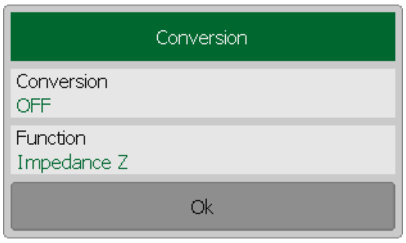
$$\frac{1}{S_{11}}$$

S-parameter complex conjugate.

S-parameter conversion function can be applied to an individual trace of a channel. Before enabling the function, first activate the trace.

To enable/disable the conversion use the following softkey **Analysis.**





Then select the **Conversion** tab and click on the **Conversion** parameter value.

To select the conversion type click on the **Function** field and select the required value from the list.

The trace format will be changed to **Lin Magnitude.**

Note	All conversion types are indicated in the trace status field, if enabled.

6.7 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 with the limit line set by the user.

The limit line can consist of one or several segments (see Figure 6.15). Each segment checks the measurement value for failing whether 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 type of the limit. The MAX or MIN limit types check if the trace falls outside the upper or lower limit respectively.

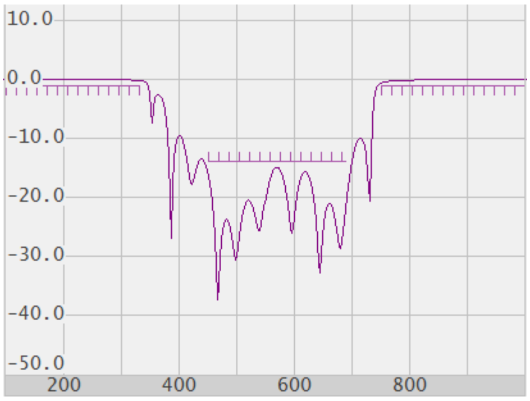


Figure 6.15 Limit line

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.

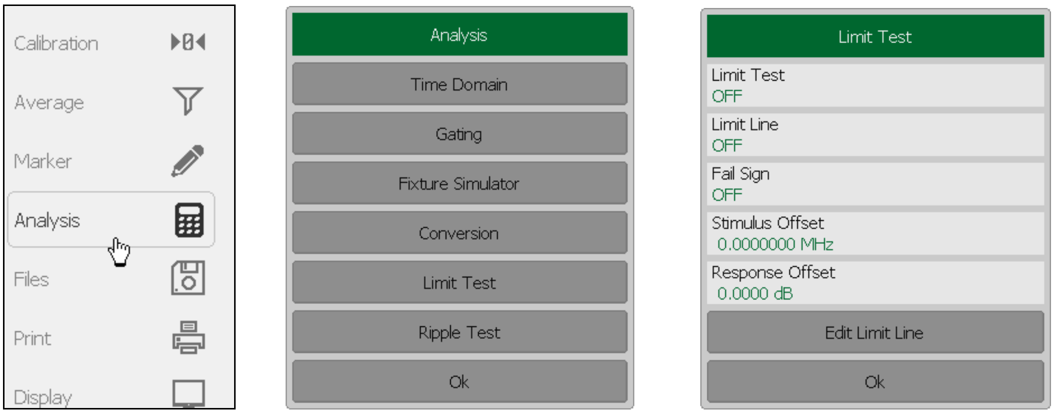
The display of the limit lines on the screen can be turned on/off independently of the status of the limit test function.

The result of the limit test is indicated in the center of the window.

If the measurement result failed **Fail** sign will be displayed in red, otherwise **Pass** sign will be displayed in green.

6.7.1 Limit Line Editing

To access the limit line editing mode use the following softkeys **Analysis > Limit Test > Edit Limit Line.**



The **Edit Limit Line** dialog will appear on the the screen (see Figure 6.16).

To add a new row in the table click **Add**. The new row will appear below the highlighted one.

To delete a row from the table click **Delete**. The highlighted row will be deleted.

To clear the entire table use **Clear Limit Table** softkey.

To save the table into *.lim file use **Save Limit Table** softkey.

To open the table from a *.lim file use **Restore Limit Table** softkey.

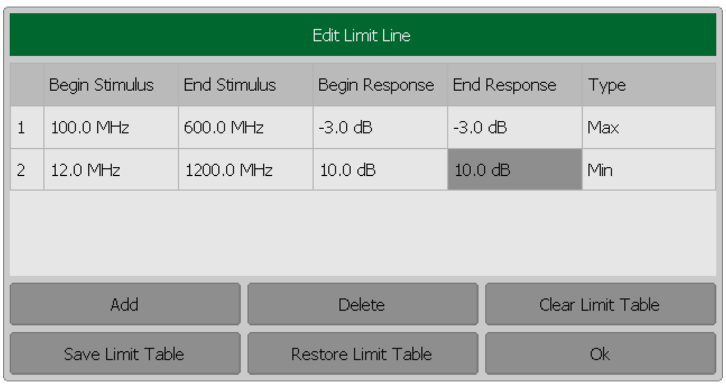


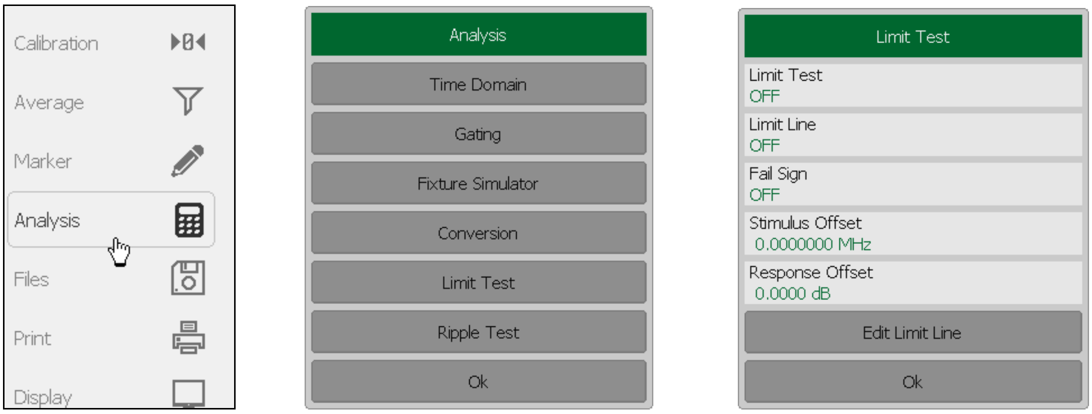
Figure 6.16 Limit line table

Navigating in the table to enter the values of the following parameters of a limit test segment:

Begin Stimulus	Stimulus value in the beginning point of the segment.
End Stimulus	Stimulus value in the ending point of the segment.
Begin Response	Response value in the beginning point of the segment.
End Response	Response value in the ending point of the segment.
Type	Select the segment type among the following: • **MAX** – upper limit; • **MIN** – lower limit; • **OFF** — segment not used for the limit test.

6.7.2 Limit Test Enabling/Disabling

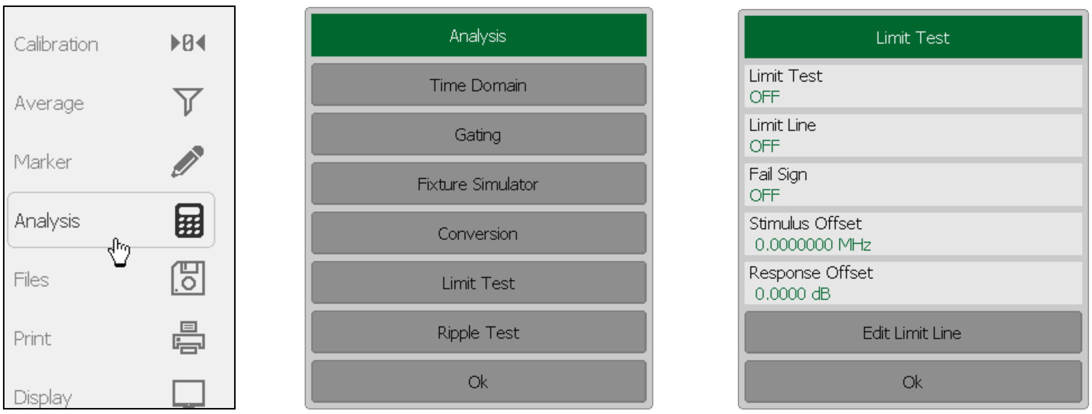
To enable/disable limit test function use the following softkeys **Analysis > Limit Test.**



Click on the **Limit Test** field to toggle between the on/off settings.

6.7.3 Limit Test Display Management

To enable/disable display of a **Limit Line** use the following softkeys **Analysis > Limit Test.**

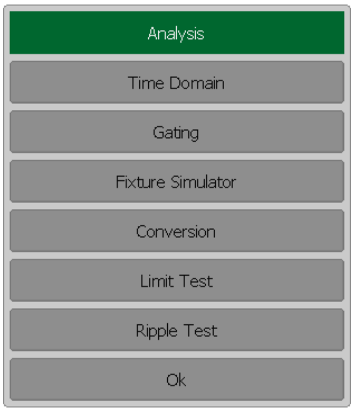


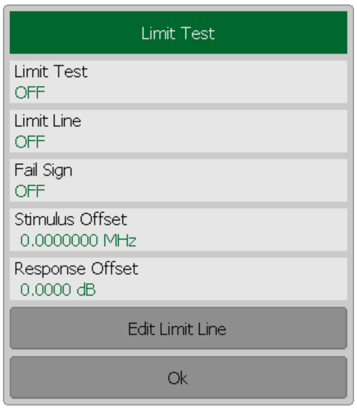
To enable/disable display of **Fail** sign in the center of the graph click on the **Limit Line** field to toggle between the on/off settings.

6.7.4 Limit Line Offset

Limit line offset function allows the user to shift the segments of the limit line by the specified value along X and Y axes simultaneously.

To define the limit line offset along X-axis use the following softkeys **Analysis > Limit Test.**





Click on the **Stimulus Offset** field and enter the value using the on-screen keypad.

To define the limit line offset along Y-axis click on the **Response Offset** field and enter the value using the on-screen keypad.

6.8 Ripple Limit Test

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**) defined by the user. 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 (see Figure 6.17). Each segment provides the ripple limit for the specific frequency band. A segment is set by the frequency band and the ripple limit value.

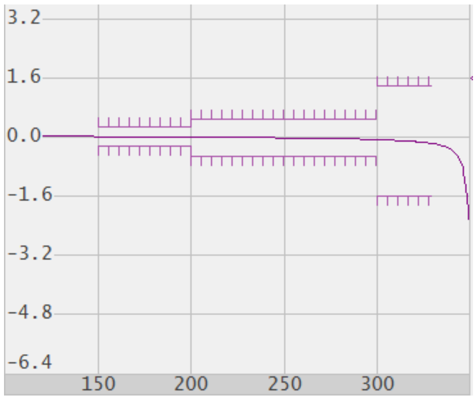


Figure 6.17 Ripple limits

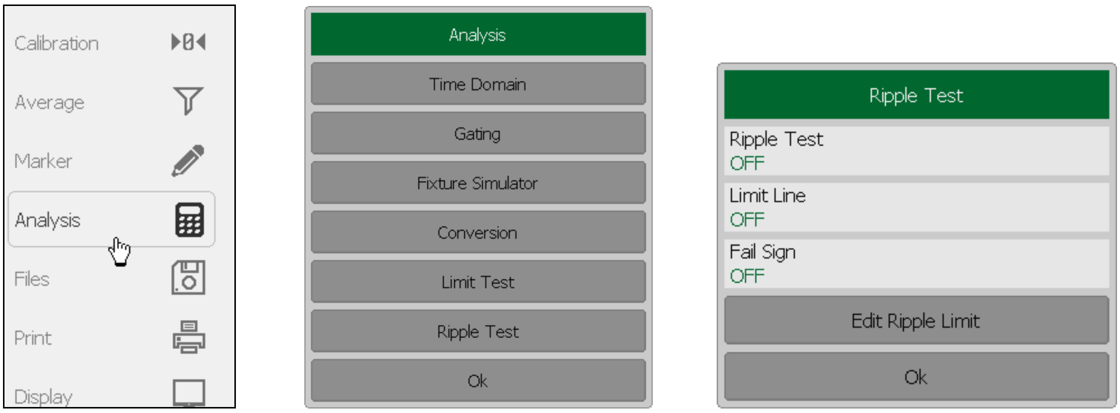
The ripple limit settings are performed in the ripple limit table. Each row of the table describes the frequency band and the ripple limit value. The ripple limit table editing is described below. The table can be saved into a *.lim file.

The display of the limit lines on the screen can be turned on/off by the user.

If the measurement result failed, **Fail** sign will be displayed in red in the center of the window.

6.8.1 Ripple Limit Editing

To access the ripple limit editing mode use the following softkeys **Analysis > Ripple Test > Edit Ripple Limit.**



The **Edit Ripple Limit** dialog will appear in the screen (see Figure 6.18).

To add a new row in the table click **Add**. The new row will appear below the highlighted one.

To delete a row from the table click **Delete**. The highlighted row will be deleted.

To clear the entire table use **Clear Ripple Limit** Table softkey.

To save the table into *.rlm file use **Save Ripple Limit** Table softkey.

To open the table from a *.rlm file use **Recall Ripple Limit** Table softkey.

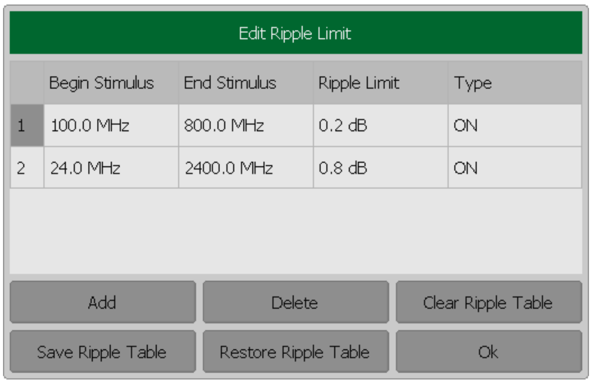


Figure 6.18 Ripple limit table

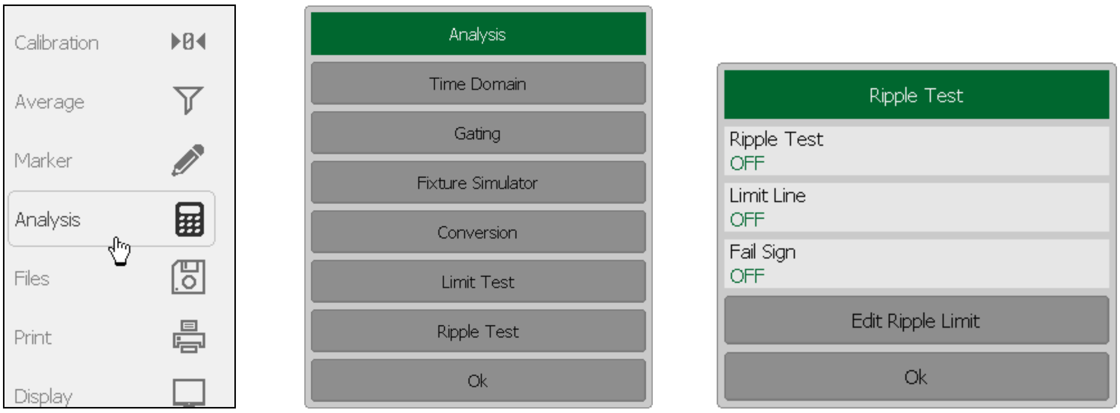
Navigating in the table to enter the values of the following parameters of a ripple limit test segment:

Begin Stimulus	Stimulus value in the beginning point of the segment.

End Stimulus	Stimulus value in the ending point of the segment.
Ripple Limit	Ripple limit value.
Type	Select the segment type among the following: • **ON** – band used for the ripple limit test; • **OFF** — band not used for the limit test.

6.8.2 Ripple Limit Enabling/Disabling

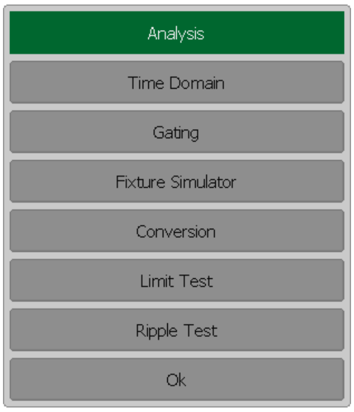
To enable/disable ripple limit test function use the following softkeys **Analysis > Ripple Test.**

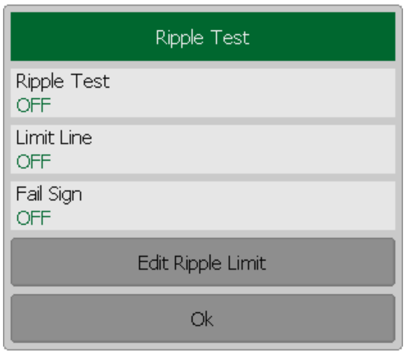


Click on the **Ripple Test** field to toggle between the on/off settings.

6.8.3 Ripple Limit Test Display Management

To enable/disable display of the ripple limit line use the following softkeys **Analysis > Ripple Test.**





Click on the **Limit Line** field to toggle between the on/off settings.

To enable/disable display of the **Fail** sign in the center of the graph use the following softkeys **Analysis > Ripple Limit.**

Click on the **Fail Sign** field to toggle between the on/off settings.

7. CABLE LOSS MEASUREMENT

While all cables have inherent loss, weather and time will deteriorate cables and cause even more energy to be absorbed by the cable. This makes less power available to be transmitted.

A deteriorated cable is not usually apparent in a *Distance to Fault* measurement, where more obvious and dramatic problems are identified. A *Cable Loss* measurement is necessary to measure the accumulated losses throughout the length of the cable.

In high-loss conditions, a *Cable Loss* measurement becomes "noisy" as the test signal becomes indistinguishable in the device noise floor. This can occur when measuring a very long cable and using relatively high measurement frequencies. To help with this condition use *High Power*, and *Averaging*.

7.1 Cable Loss Measurement Algorithm

In order to measure Cable Loss, perform the following steps:

- Set the device to initial state using the buttons **System > Preset**;
- Select for the current trace type of measurement **Cable Loss**;
- Set the Start and Stop frequency of measurements;
- Perform a full 1-port calibration for measuring port;
- Connect the cable to be tested;
- Connect a LOAD at the end of the cable to be tested. This limits the reflections to faults that are located in the cable under test. These reflections are visible on the screen as "ripple" or low-level standing waves and obscure the actual loss of the cable;
- Save the trace data in memory using the buttons **Trace -> Memory Trace;**
- Remove the LOAD and leave the end of the cable to be tested open;
- Press **Trace > Data Math > Data – Mem** The ripple in the measurement is removed. These minor imperfections in the cable should not be considered in the **Cable Loss** measurement;
- Use Averaging to remove random noise from high-loss measurements. To turn on the averaging press the buttons **Average > Averaging.**

The displayed trace shows the Cable Loss values in one direction through the cable. A Return Loss measurement would show the loss for both down the cable and back.

In the current example you can see the cable loss for 30-meter coaxial cable with loss parameters 0.397dB/m at 1GHz frequency (see Figure 7.1).

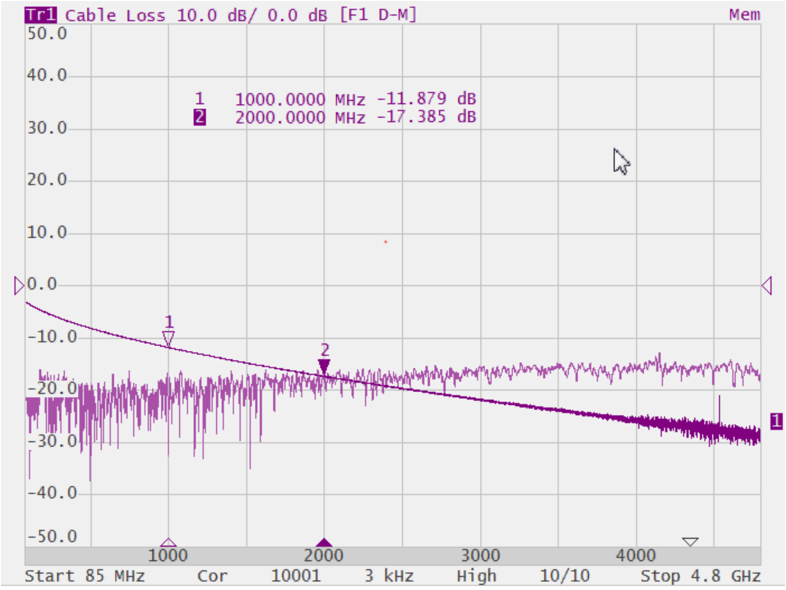


Figure 7.1 Cable loss measurement

8. ANALYZER DATA OUTPUT

8.1 Analyzer State

The Analyzer state, calibration, actual trace and memory traces can be saved to the Analyzer state file and later uploaded back into the Analyzer program.

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
All	The Analyzer settings, table of calibration coefficients, and data traces

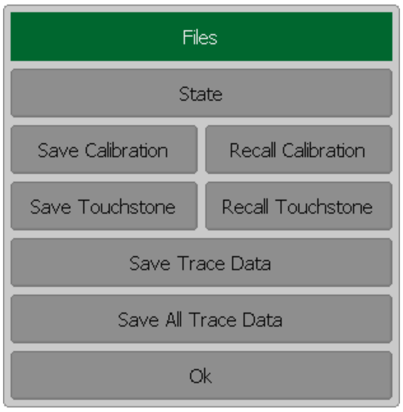
The Analyzer settings that become saved into the state file are the parameters, which can be set in the following submenus of the softkey menu:

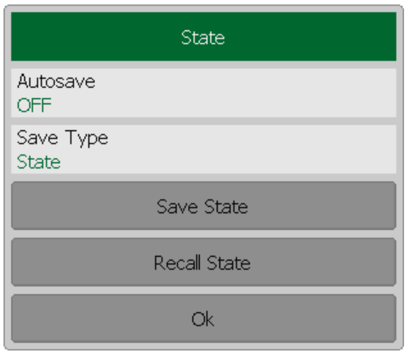
- All the parameters in Stimulus submenu;
- All the parameters in Scale submenu;
- All the parameters in Channel submenu;
- All the parameters in Trace submenu;
- All the parameters in System submenu;
- All the parameters in Average submenu;
- All the parameters of Markers submenu;
- All the parameters of Analysis submenu;

A special **Autosave.cfg** file is used to recall automatically the Analyzer state after start. To be able to use this function, you have to enable the automatic state saving mode.

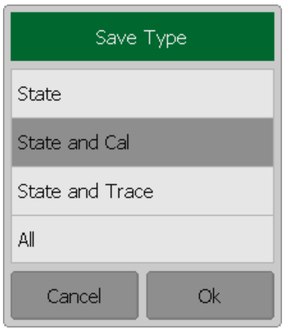
8.1.1 Analyzer State Saving

To save the Analyzer state use the following softkeys **Files > State > Save Sate.**





To set type of saving click on **Save Type** field. Select type in **Save Type** dialog and click **Ok**. Select a path and enter the state file name in the pop-up dialog.



Navigation in directory tree is available in **Save State** dialog.

To open a directory and activate it, double click on the directory name.

To go up in the directory hierarchy, double click on the ".." field.

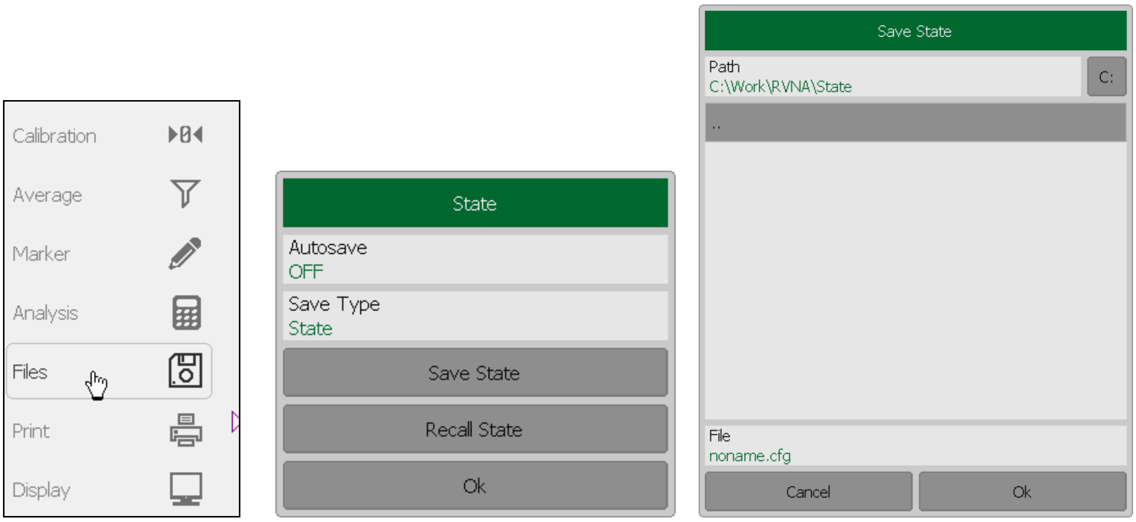
To select the disk click **Drive.**

To change the name of the saved state file using the on-screen keypad click on the **File** field.

To save the state file in the **Save State** dialog click **Ok.**

8.1.2 Analyzer State Recalling

To recall the state from a file of Analyzer state use the following softkeys **Files > State > Recall State.**



Select the state file name in the pop-up dialog.

Navigation in directory tree is available in **Recall State** dialog.

To open a directory and activate it, double click on the directory name.

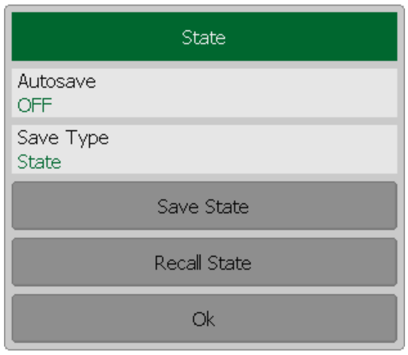
To go up in the directory hierarchy, double click on the "**...**" field.

To select the disk click **Drive.**

To recall the state from file in the **Recall State** dialog click **Ok.**

8.1.3 Autosave and Autorecall State of Analyzer

To save a state which will be automatically recalled after start use the softkey **Files > State.**



Click in the **Autosave** parameter value field. The parameter value will change to **ON**.

When exiting, the state will be saved. The next time the program state will be restored.

8.2 Channel State

A channel state can be saved into the RAM. The channel state saving procedure is similar to the Analyzer state saving and the same saving types (described in section 8.1.1) are applied to the channel state saving.

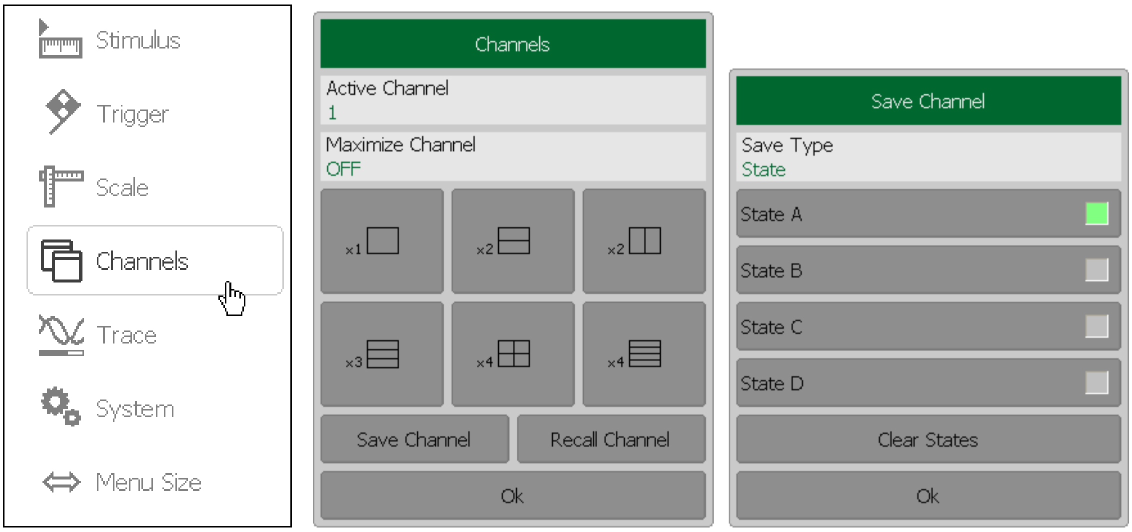
Unlike the Analyzer state, the channel state is saved into the RAM (not to the hard disk) and is cleared when the program is closed.

For channel state storage, there are four memory registers A, B, C, D.

The channel state saving allows the user to easily copy the settings of one channel to another one.

8.2.1 Channel State Saving

To save the Channel state use the following softkeys **Channels > Save Channel.**



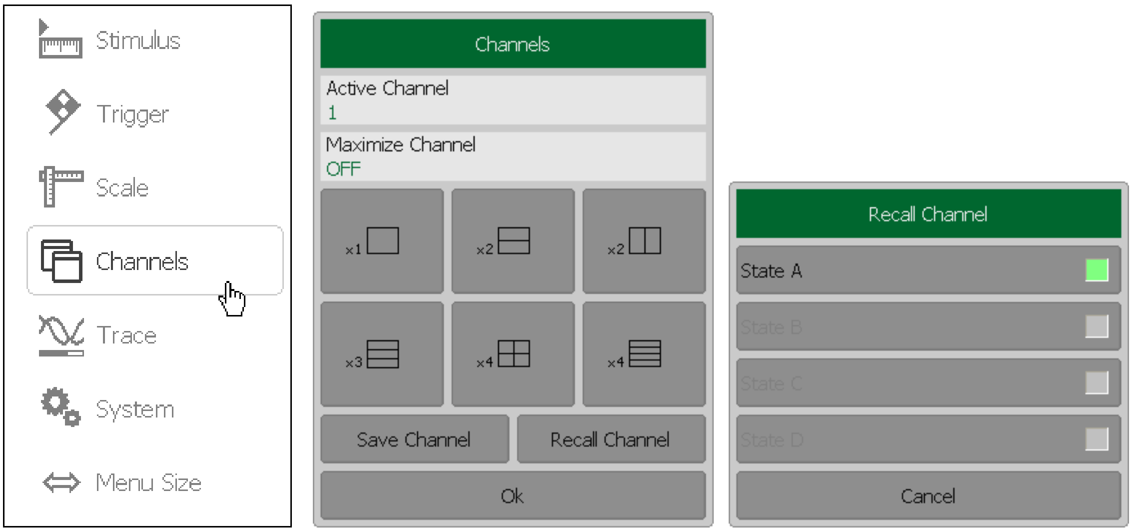
To save the state click **State A | State B | State C | State D** softkey in the **Save Channel** dialog.

To select a save option click on **Save Type** field.

To clear all saved states click on **Clear States** softkey.

8.2.2 Channel State Recalling

To recall the active channel state use the following softkeys **Channels > Recall Channel.**



Click the required softkey of the available **State A | State B | State C | State D.**

If the state with some number was not saved the corresponding softkey will be grayed out.

8.3 Trace Data CSV File

The Analyzer allows to save an individual trace data as a CSV file (comma separated values). The *.CSV file contains digital data separated by commas. The active trace stimulus and response values in current format are saved to *.CSV file.

Only one (active) trace data are saved to the file.

The trace data are saved to *.CSV in the following format:

F[0],	**Data1,**	**Data2**
F[1],	**Data1,**	**Data2**
	. . .	
F[N],	**Data1,**	**Data2**

F[n] – frequency at measurement point n;

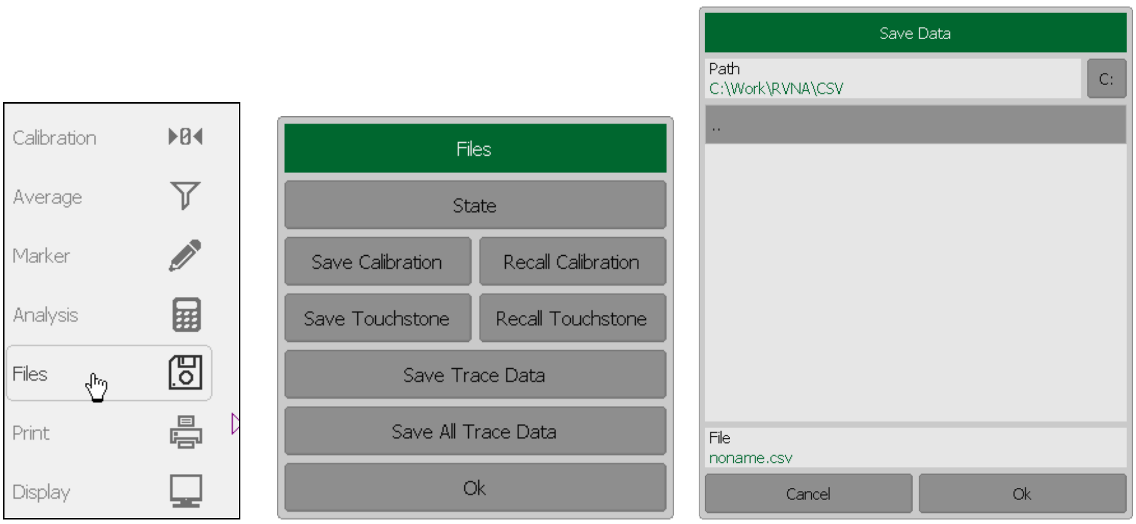
Data1 – trace response in rectangular format, real part in Smith chart and polar format;

Data2 – zero in rectangular format, imaginary part in Smith chart and polar format.

8.3.1 CSV File Saving

To save the trace data, first activate the trace.

To save the trace data use the following softkeys **Files > Save Trace Data.**



Select a path and enter the file name in the pop-up dialog.

Navigation in directory tree is available in **Save Data** dialog.

To open a directory and activate it, double click on the directory name.

To go up in the directory hierarchy, double click on the "**...**" field.

To select the disk click the disk letter softkey.

To change the name of the saved file using the on-screen keypad, double click on the **File** field.

To save the file, in the **Save Data** dialog click **Ok.**

8.4 Trace Data Touchstone File

The Analyzer allows the user to save S-parameters to a Touchstone file. The Touchstone file contains the frequency values and S-parameters. The files of this format are typical for most of circuit simulator programs.

The *.s1p files are used for saving the parameters of a 1-port device.

The *.s2p files are used for saving the parameters of a 2-port device.

Only one active trace data are saved to the file.

The Touchstone file contains comments, header, and trace data lines. Comments start with «!» symbol. Header starts with «#» symbol.

The *.s1p Touchstone file for 1-port measurements:

! Comments		
# Hz S FMT R Z0		
F[1]	**{S_{11}}'**	**{S_{11}}"**
F[2]	**{S_{11}}'**	**{S_{11}}"**
	. . .	
F[N]	**{S_{11}}'**	**{S_{11}}"**

The *.s2p Touchstone file for 2-port measurements:

! Comments								
# Hz S FMT R Z0								
F[1]	**{S_{11}}'**	**{S_{11}}"**	**{S_{21}}'**	**{S_{21}}"**	**{S_{12}}'**	**{S_{12}}"**	**{S_{22}}'**	**{S_{22}}"**
F[2]	**{S_{11}}'**	**{S_{11}}"**	**{S_{21}}'**	**{S_{21}}"**	**{S_{12}}'**	**{S_{12}}"**	**{S_{22}}'**	**{S_{22}}"**
	. . .		. . .		. . .		. . .	
F[N]	**{S_{11}}'**	**{S_{11}}"**	**{S_{21}}'**	**{S_{21}}"**	**{S_{12}}'**	**{S_{12}}"**	**{S_{22}}'**	**{S_{22}}"**

where:

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

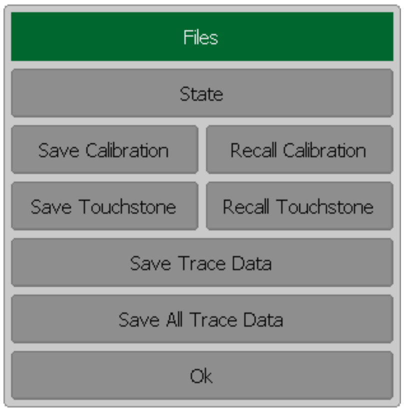
{...}' – {real part (RI) | linear magnitude (MA) | logarithmic magnitude (DB)}

{...}" – {imaginary part (RI) | phase in degrees (MA) | phase in degrees (DB)}

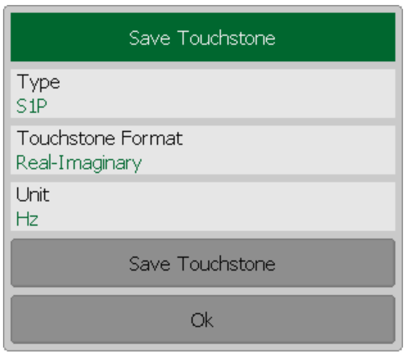
The Touchstone file saving function is applied to individual channels. To use this function, first activate the channel.

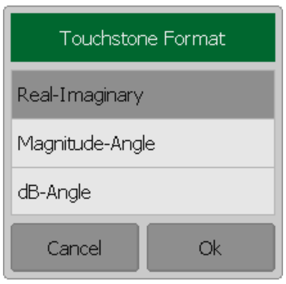
8.4.1 Touchstone File Saving

To save the Touchstone format data use the following softkeys **Files > Save Touchstone.**



To select the saved Touchstone file format click on the **Touchstone Format** field and select the required format from the **Touchstone Format** list.





Complete by **Ok.**

To select the type (S1P or S2P) of Touchstone file click on the **Type** field.

Actual data is used for S11 and zero values for S12, S21, S22.

Click **Save Touchstone** softkey.

Select a path and enter the file name in the pop-up dialog.

Navigation in directory tree is available in **Save Touchstone** dialog.

To open directory and activate it, double click on the directory name.

To go up in the directory hierarchy, double click on the "**...**" field.

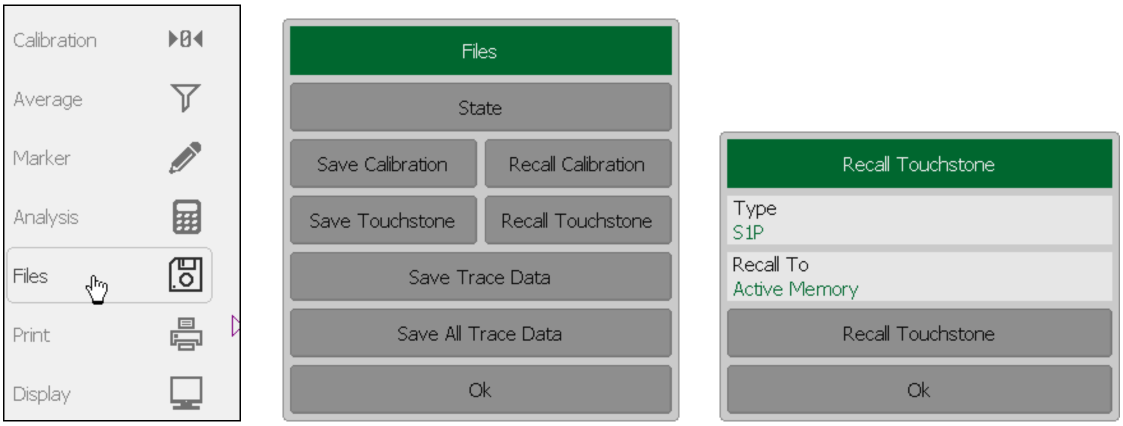
To select the disk click the disk letter softkey.

To change the name of the saved file using the on-screen keypad click on the **File** field.

To save the file, in the **Save Touchstone** dialog click **Ok.**

8.4.2 Touchstone File Recalling

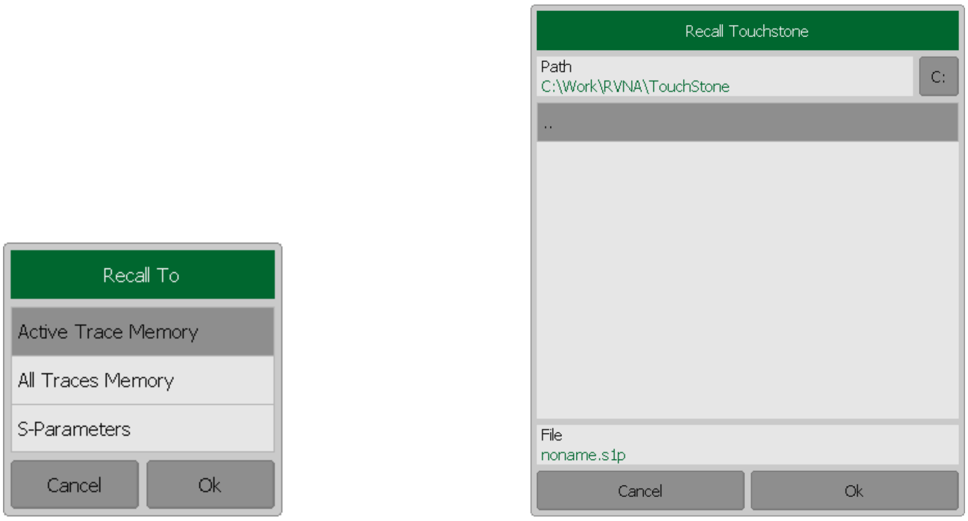
To recall the data trace use the following softkeys **Files > Recall Touchstone.**



You can load data into the active trace memory, all trace memory or measured by the S parameter.

To select download option click on the **Recall To** field.

Complete by **Ok.**



Select a path and enter the file name in the pop-up dialog.

Navigation in directory tree is available in **Recall Touchstone** dialog.

To open directory and activate it, double click on the directory name.

To go up in the directory hierarchy, double click on the "**...**" field.

To select the disk click the disk letter softkey.

To recall the file in the **Recall Touchstone** dialog click **Ok.**

Note	After downloading the file touchstone in the S-parameters frequency scanning will stop

8.5 Graph Printing

This section describes the print/save procedures for the graph data.

You can print out the graphs using three different applications:

- MS Word;
- Image Viewer for MS Windows;
- Save screen shot in *.png format using the program menu

Note	MS Word application must be installed in MS Windows system.

You can select the print color before the image is transferred to the printing application:

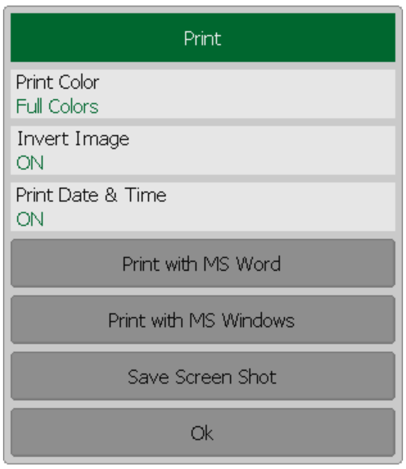
- Color (no changes);
- Gray Scale;
- Black & White.

You can invert the image before it is transferred to the printing application.

You can add current date and time before the image is transferred to the printing application.

8.5.1 Graph Printing Procedure

To print channels graph area use the following softkeys **Print > Print with MS Word | Print with MS Window.**



To select the print color click on the **Print Color** field.

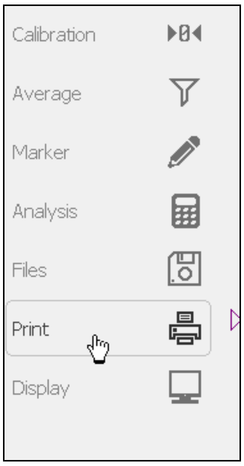
If necessary, invert the image by **Invert Image** field.

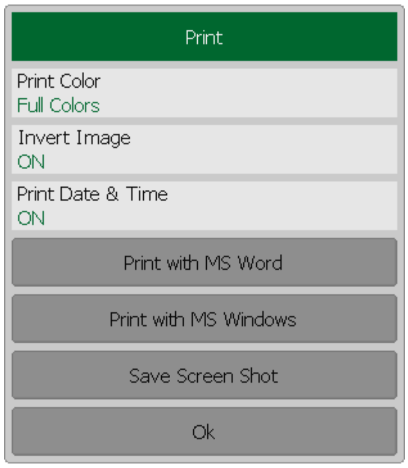
If necessary, select printing of date and time by **Print Date& Time** field.

Close **Print** dialog by **Ok.**

8.5.2 Quick saving program screen shot

To save screen shot of the channels graph data use the **Print** softkey.





Click **Screen Shot** softkey in the **Print** dialog.

The files will be saved to the **Image** folder located in the main program folder. The saved files will be automatically assigned the following name: **scrXXXXX.png** where XXXXX is automatically incremented ordinal number.

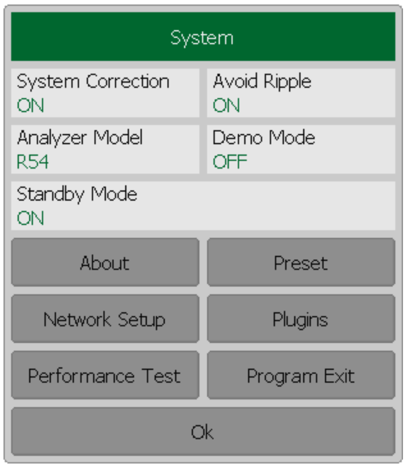
9. SYSTEM SETTINGS

9.1 Analyzer Presetting

Analyzer presetting feature allows the user to restore the default settings of the instrument.

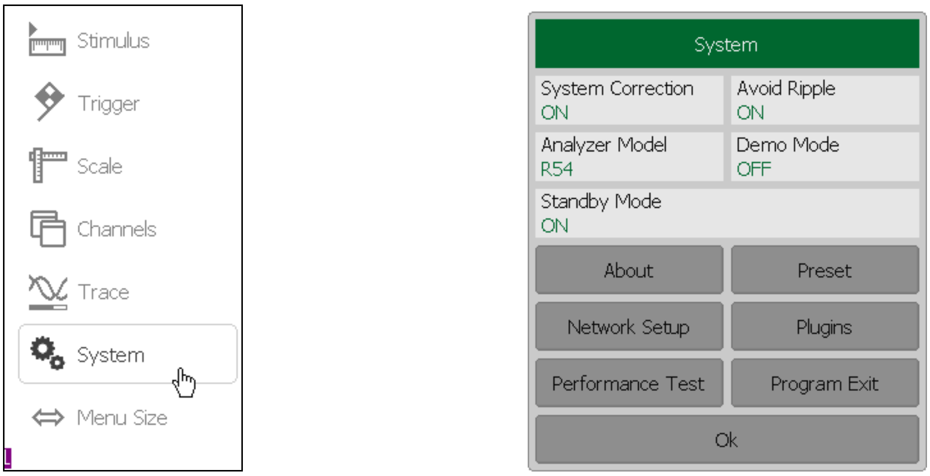
The default settings of your Analyzer are specified in Appendix 1.

To preset the Analyzer use the following softkeys **System > Preset.**



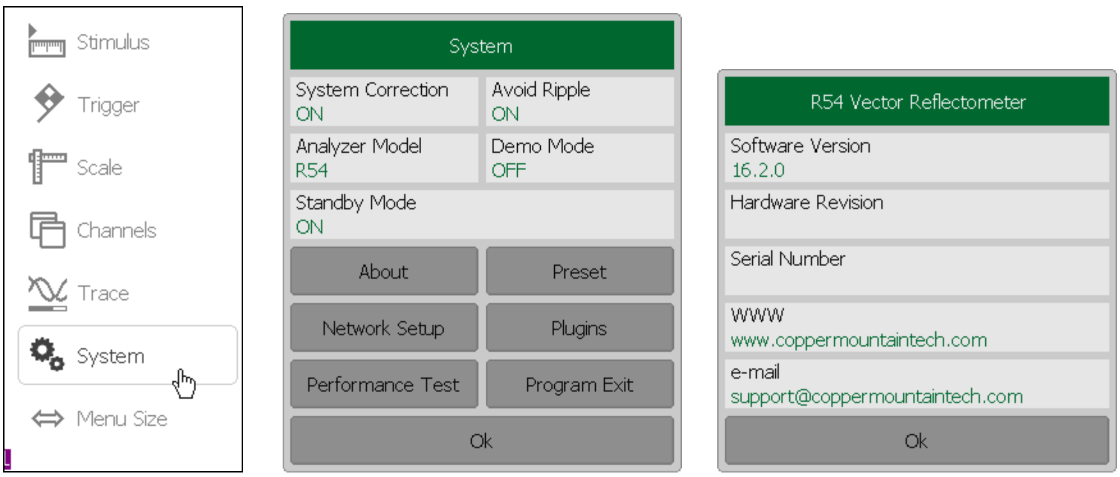
9.2 Program Exit

To exit the program use the following softkeys in the right menu bar **System > Program Exit.**



9.3 Analyzer System Data

To get the information about software version, hardware revision and serial number of the Analyzer use the following softkeys in the right menu bar **System > About.**



9.4 System Correction Setting

The Analyzer is supplied from the manufacturer calibrated with the calibration coefficients stored in its non-volatile memory. The factory calibration is used by default for initial correction of the measured S-parameters. Such calibration is referred to as system calibration, and the error correction is referred to as system correction.

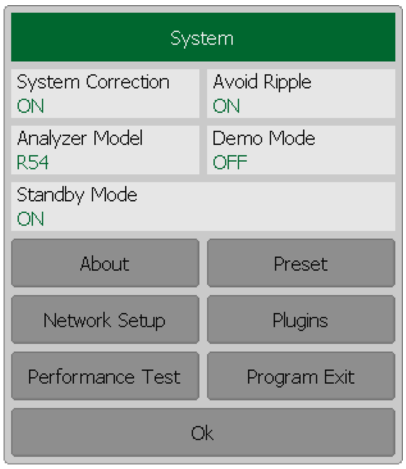
The system correction ensures initial values of the measured S-parameters before the Analyzer is calibrated by the user. The system calibration is performed at the plane of the port physical connectors and leaves out of account the cables and other fixture used to connect the DUT. The measurement accuracy of the Analyzer without its calibration with the user setup is not rated.

Normally, the disabling of the system correction **is not required** for a calibration and further measurements.

The system correction can be disabled only in case the user provided a proper calibration for the Analyzer. The measurement accuracy is determined by user calibration and does not depend on the system correction status. The only rule that should be observed is to disable/enable the system correction before the user calibration, so that the calibration and further measurement could be performed under the same conditions.

If the system correction is disabled by the user, this is indicated in the instrument status bar:

To disable/enable the system correction use the **System softkey.**



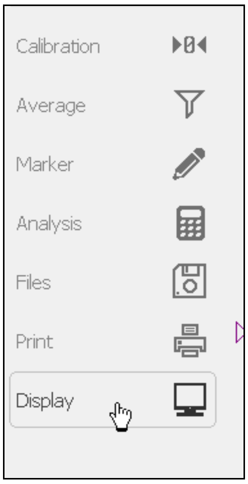
Click on the **System Correction** field to toggle between the on/off settings.

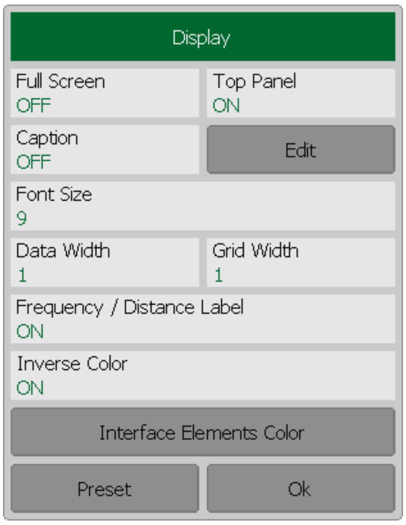
9.5 User Interface Setting

The Analyzer enables you to make the following user interface settings:

- Toggle between full screen and window display;
- Width of traces;
- Font size in channel window;
- Inverting colors in graph area;
- Show/hide the channel title bar.

To toggle between full screen and window display use the following softkey **Display.**





Click on **Full Screen** field to change the parameter value.

To change the data width and grid width click on **Data Width** and **Grid Width** fields respectively and enter the required value using the on-screen keypad.

The width can be set from 1 to 4.

The changes made to the width of the data and grid will affect all the channels.

To change the font size in the channel window click on **Font Size** field and enter the required value using the on-screen keypad.

The size can be set from 8 to 24.

To change the color of the background of the graph click on **Inverse Color** field to toggle between the on/off settings.

To show/hide the channel title bar click on **Caption** field in the pop-up dialog to toggle between the ON/OFF settings.

To restore the default factory settings use the softkeys **Display > Preset.**

10. SPECIFICS OF WORKING WITH TWO OR MORE DEVICES

Additional software for devices allows to use simultaneously up to eight devices. This expands the list of parameters to be measured. You can measure Scalar transfer coefficient in two directions, for example $|S_{21}|$ and $|S_{12}|$ of the DUT.

The signal source can be only one device (active). The rest devices (passive) work as a signal receiver. Active device has a green indicator READY/STANDBY, which is located on the top cover. The passive device has at the same time red and green LEDs.

Active instrument is assigned according to the measured S-parameters. For example when measuring the parameters S_{11} and S_{21} the first device will be an active one, when measuring S_{12} and S_{22} - the second one. If the channel window has a list of the S-parameters, the program will make a few launches of the scanning.

10.1 Installation of additional software

For simultaneous work with several analyzers you need a program RNVNAx8.exe. The installation file is called Setup_RNVNAx8.exe _vX.X.exe. XX -it is a version of the software. Installation procedure is similar to that described in paragraph 2.2.

10.2 Connecting devices to a USB port

When running the software with several devices, each of them is assigned a port number in the order of their connection to the personal computer. If the analyzers were connected to the USB interfaces of the computer before starting the program, the numbering of the ports will follow the internal numbering of the USB host interfaces.

Important! If devices are supposed to use the synchronization mode via the USB bus, then all analyzers must be connected to USB interfaces that are serviced by one controller. Usually, this is a nearby USB port of a personal computer. If, when using USB bus synchronization, the analyzers are connected to different USB controllers within the same computer, the devices can not be synchronized. A good solution is to use an external USB HUB with its own power supply.

10.3 Synchronizing the work of analyzers

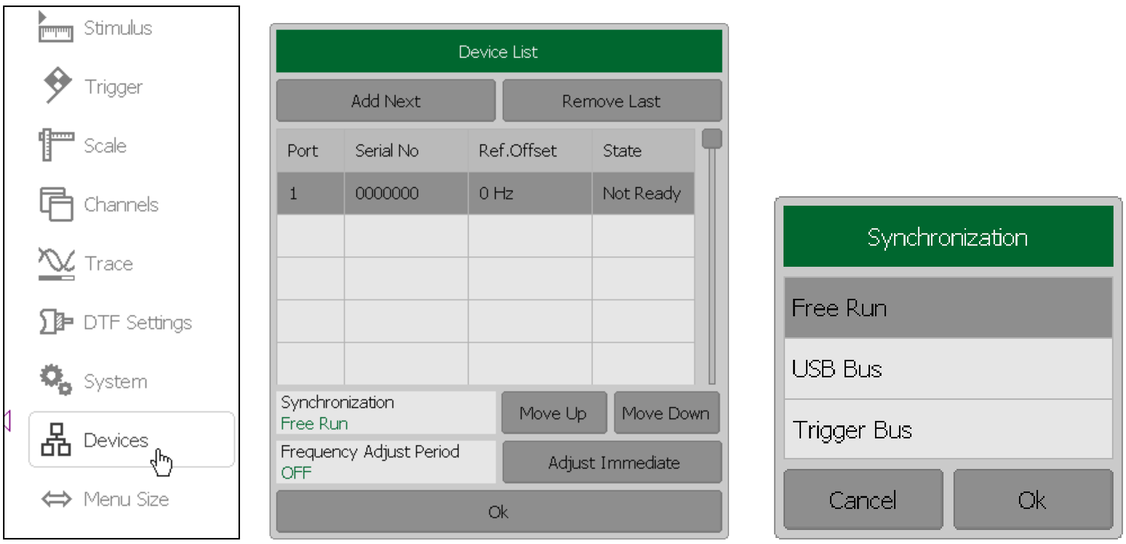
To perform the measurement of the transmission factors between the devices, it is necessary to synchronize their operation. If the task of measuring the transmission factors is not set and independent circuits are measured, where the signal source of one device can not be a difficalty to the operation of another analyzer, you can use the device free run mode, that is, operate without synchronization.

The software allows you to select the following options for the operation of devices:

- Free Run;
- USB Bus;
- Trigger Bus.

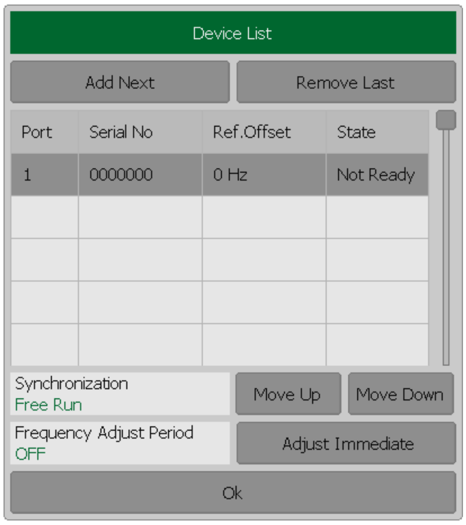
To synchronize the work on the trigger bus, it is necessary to connect the inputs / outputs of the external trigger of every device to each other with a coaxial cable.

To choose the nessesary type of synchronization press the following soft keys **Devices > Synchronization > Free Run | USB Bus | Trigger Bus.**



10.4 Adding / removing devices

To add or remove a device press rhe following softkeys **Devices > Add Next | Remove Last.**



10.5 Frequency adjustment of the internal generators

Internal reference generators of analyzers have the finite accuracy of frequency. When working with several devices you need to set the output frequency of each of them relatively to the first device in the list. This eliminates the error in the measurement of the transmission coefficients, which arises from the fact that the frequency of a single device does not fall in the bandwidth of the filter of another device.

By default, when you connect the devices function of automatic frequency starts to work. The parameters of the automatic adjustment and its periodicity can be specified by user.

When performing the frequency adjustment ports of the analyzers should be connected between themselves. It is necessary to ensure the weakening of the signal between the ports is not more than 50 dB.

The program provides auto tuning of frequency on the central frequency range, which is used in the active channel.

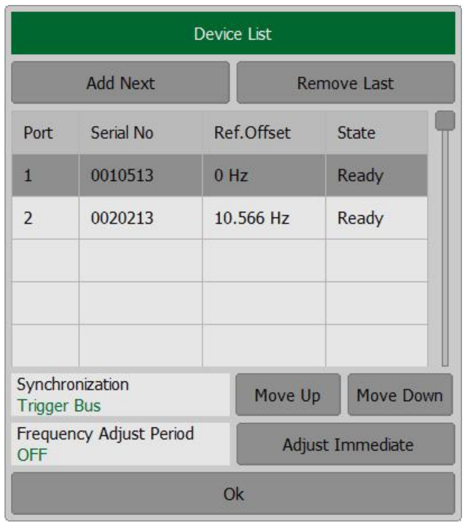
Before using the analyzers should be warmed up, to minimize the temperature drift of reference generators.

When a reference frequency source is selected **Linked** the analyzer uses a common reference frequency bus, in this case the first device is the source, the remaining devices are the receivers.

Note	If the reference frequencies of the two analyzers are connected to each other by a coaxial cable and the source of the reference frequency is selected, either **External** or **Linked** frequency tuning is not required. The value of the frequency tuning in this case is taken to be zero

10.5.1 Manual frequency adjustment

To perform manual frequency adjustment press the soft keys: **Devices > Adjust Immediate.**

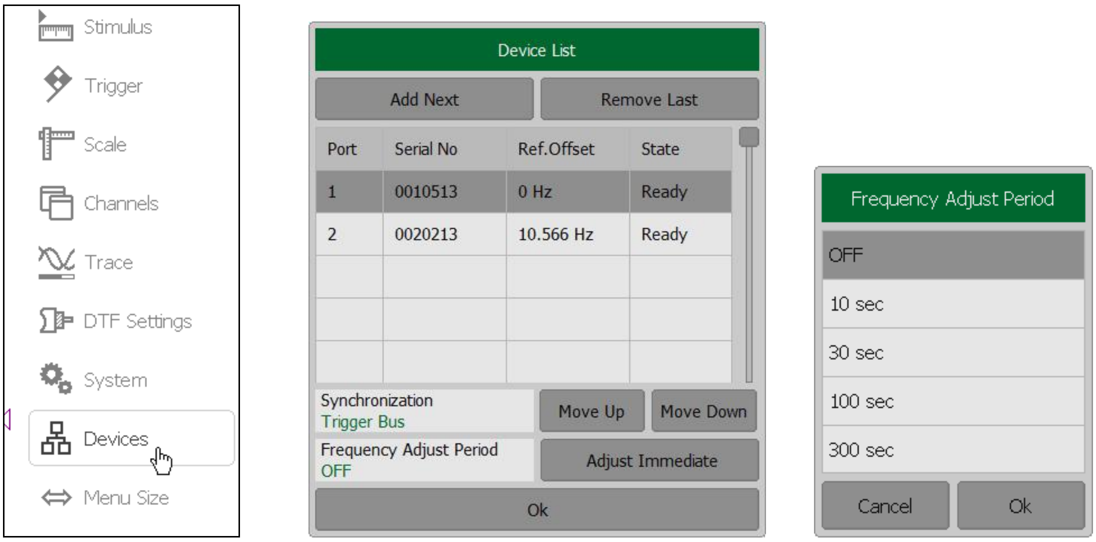


After the adjustment in the field **Ref. Offset** will be shown a correction of the reference oscillator frequency of the second device.

10.5.2 Automatic frequency adjustment

In the automatic adjustment mode the program performs the adjustment after a specified time interval. The real interval of the djustment can be more than specified.

To perform automatic frequency adjustments press the softkey **Devices.** Click the left mouse button on the field **Frequency Adjust Period.**



In dialogue form **Frequency Adjust Period** select the time interval and press **OK**.

10.6 Features of analyzers calibration

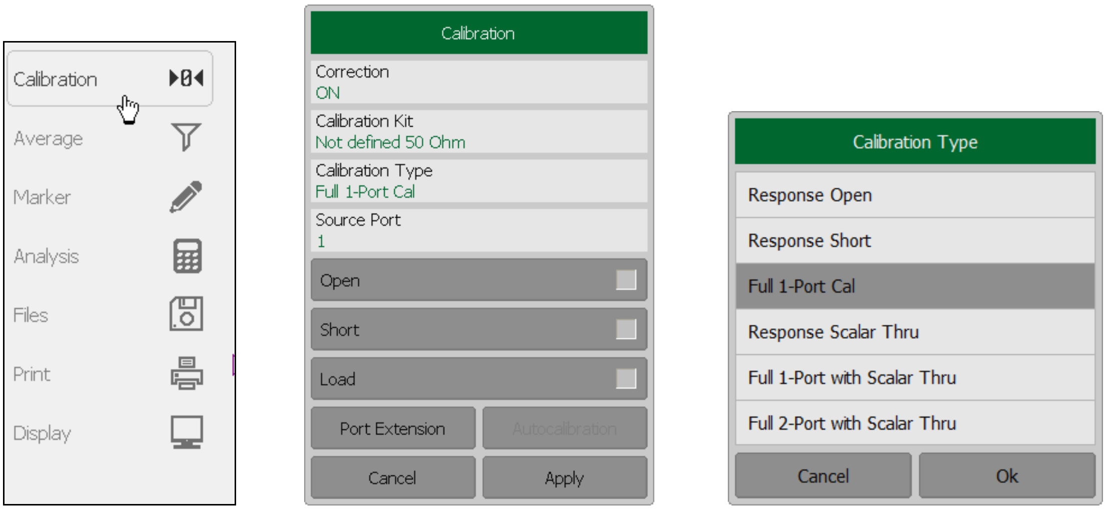
The calibration procedure described in paragraph 5, is expanded by selection of ports, and the ability to calibrate the scalar coefficient of transmission. Before calibration of **THRU** the adjustment of frequency generators will be executed.

Table 10.1 Error correction field

Symbols	Definition
ST	Transmission normalization
F1ST	Full 1port Calibration with transmission normalization
F2ST	Full 2port Calibration with transmission normalization
MATH	Equivalent to F2ST Calibration, obtained by mathematical method.

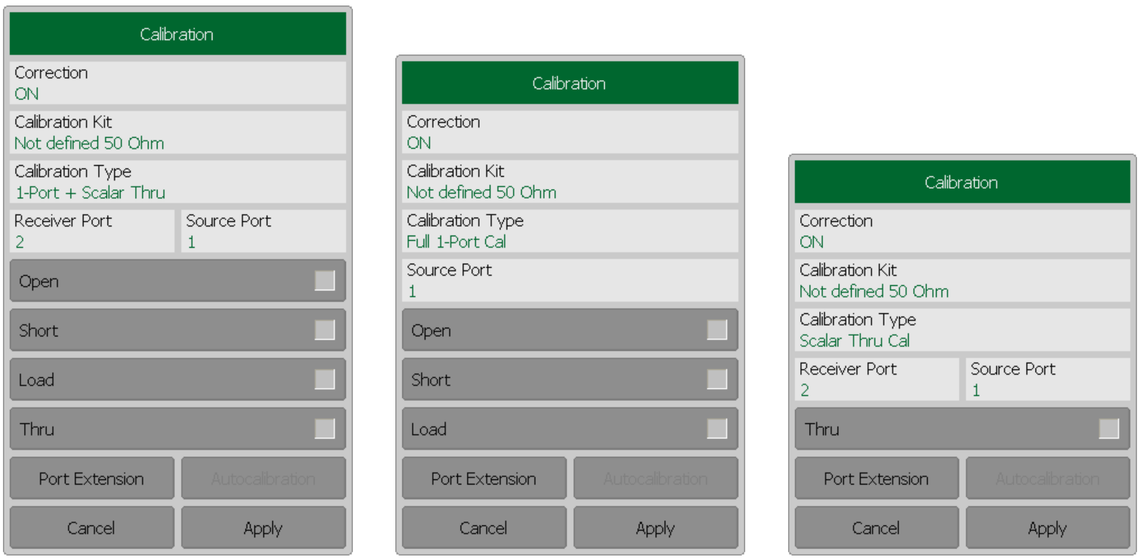
10.6.1 Calibration Type

To select the ports, use the following softkey **Calibration**.



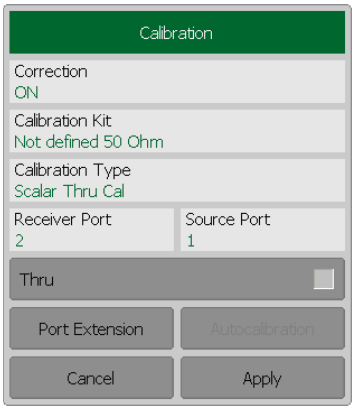
Click on the **Source Port** field to select required ports from the list. Then click on the **Calibration Type** field.

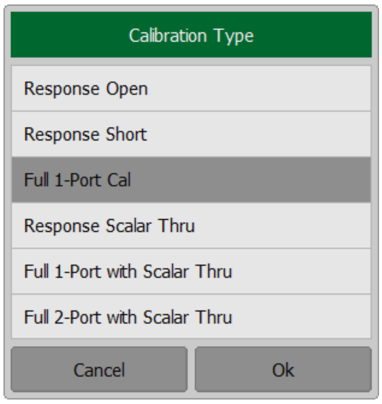
Depending on the type of calibration, the choice of source port or source port and signal receiver port will be available.



10.6.2 Scalar Transmission Normalization

To execute transmission normalization use **Calibration** softkey.





Then click on the **Calibration Type** field. In the dialogue form **Calibration Type** choose **Response Scalar Thru**. Complete the setting by **Ok**

In the dialogue form **Calibration** assign a signal source port and a signal receiver port.

Connect the analyzers ports by **Thru** standard.

Press the softkey **Thru** and wait until the measurement is complete.

To complete the calibration procedure, click **Apply**. This will activate the process of calibration coefficient table calculation and saving it into the memory. The error correction function will also be automatically enabled. To clear the measurement results of the standard, click **Cancel**.

Note	You can check the calibration status in trace status field (See section 4.2.2).

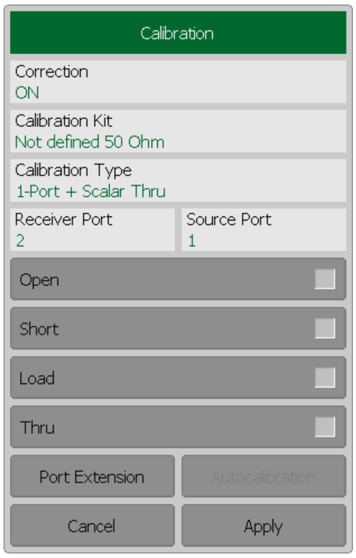
10.6.3 Expanded Scalar Transmission Normalization

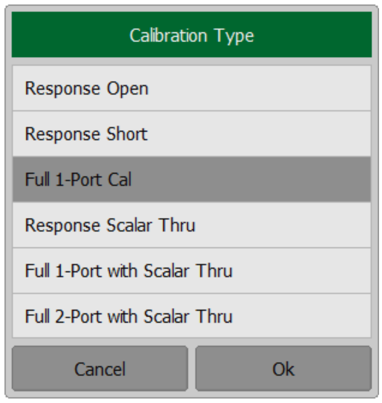
The extended normalization of the transmission coefficient module is characterized by the presence of a full one-port calibration of the source port (F1ST) or full one port calibrations of the source and receiver ports (F2ST). This makes it possible to increase the accuracy of the transmission coefficient measurements by taking into account the matching of the signal source to the measured device.

In the description of the calibration procedure below, we will write about the choice of F1ST - full one-port calibration with normalization of the transmission coefficient module or F2ST - full two-port calibration with normalization of the transmission coefficient module.

Before starting calibration perform the following settings: select active channel, set the parameters of the channel (frequency range, IF bandwidth, etc), and select the calibration kit.

To execute scalar transmission normalization F1ST use **Calibration** softkey.





Then click on the **Calibration Type** field. In the dialogue form **Calibration Type** choose **Full 1-Port with Scalar Thru** . Complete the setting by **Ok.**

In the dialogue form **Calibration** assign a signal source port and a signal receiver port.

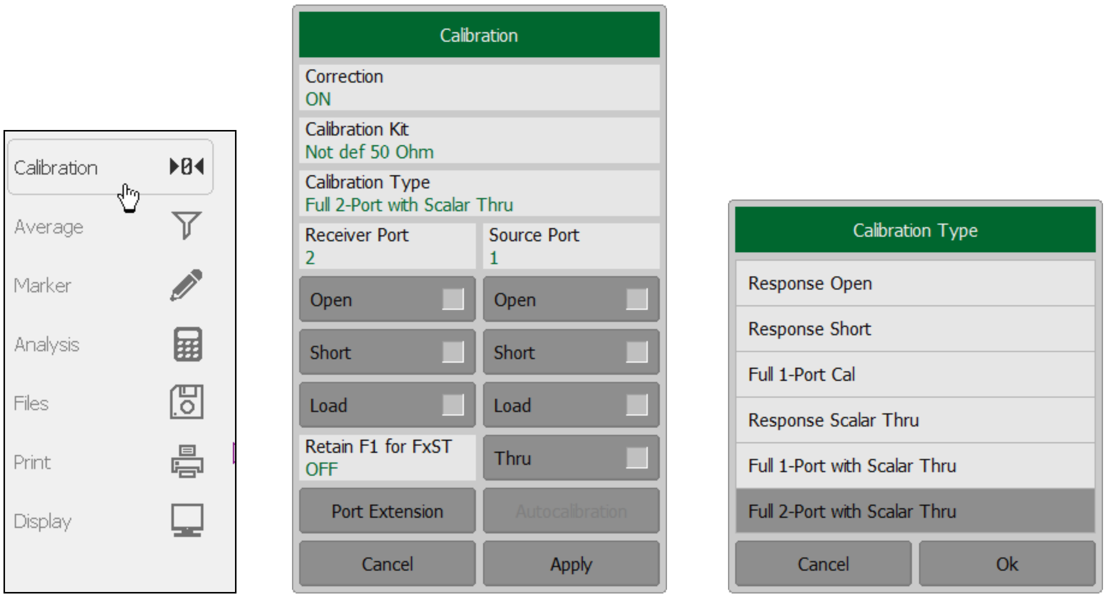
Connect the analyzers ports by **Thru** standard.

Press the softkey **Thru** and wait until the measurement is complete.

Connect **Open, Short, Load** standards to the source port in any order. Perform measurements, pressing the softkeys **Open**, **Short** or **Load** respectively.

To complete the calibration procedure, click **Apply**. To clear the measurement results of the standard, click **Cancel**.

To execute scalar transmission normalization F2ST use **Calibration** softkey.



Then click on the **Calibration Type** field. In the dialogue form **Calibration Type** choose **Full 2-Port with Scalar Thru** . Complete the setting by **Ok.**

In the dialogue form **Calibration** assign a signal source port and a signal receiver port.

Connect the analyzers ports by **Thru** standard.

Press the softkey **Thru** and wait until the measurement is complete.

Connect **Open, Short, Load** standards to the source port in any order. Perform measurements, pressing the softkeys **Open**, **Short** or **Load** respectively.

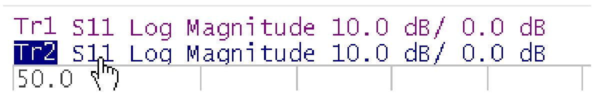
Connect **Open, Short, Load** standards to the receiver port in any order. Perform measurements, pressing the softkeys **Open**, **Short** or **Load** respectively.

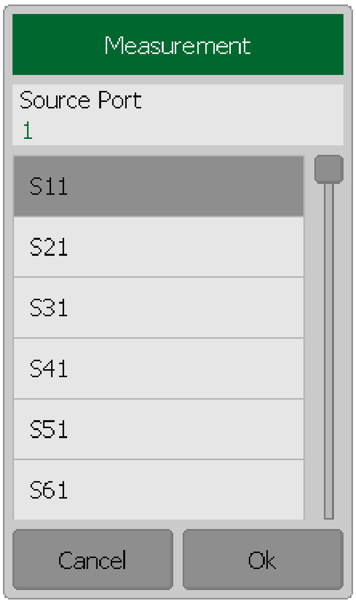
To complete the calibration procedure, click **Apply**. To clear the measurement results of the standard, click **Cancel**.

10.7 Selection of the measured S-parameters

A measured parameter (S_{11}, S_{21}, S_{12} , S_{22} etc) is set for each trace. Before you select the measured parameter, first activate the trace.

To assign the measured parameters to a trace, make a mouse click on the S-parameter name in the trace status line and select the required parameter in the dialog **Measurement**.





Complete the setting by **Ok**.

---

11. MAINTENANCE AND STORAGE

11.1 Maintenance Procedures

This section describes the guidelines and procedures of maintenance, which will ensure fault-free operation of your Analyzer.

The maintenance of the Analyzer consists in cleaning of the instrument, factory calibrations, and regular performance tests.

11.2 Instrument Cleaning

This section provides the cleaning instructions required for maintaining the proper operation of your Analyzer.

To remove contamination from parts other than test ports and 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 keep the test ports always 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:

- 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).

Always dry a connector completely before using it.

Never use water or abrasives for cleaning any connectors of the Analyzer.

Do not allow contact of alcohol to the insulators surface of the connectors.

When connecting male-female coaxial connectors always use a calibrated wrench.

11.3 Factory Calibration

Factory calibration is a regular calibration performed by the manufacturer or an authorized service center. We recommend you to send your Analyzer for factory calibration every three years.

Factory calibration is a full one-port Analyzer calibration. It can be performed in two following modes: with high output power and with low output power. The calibration coefficients employed during the Analyzer operation correspond to the selected mode of the output power.

The factory calibration of the Analyzer allows performing measurement without additional calibration and reduces the measurement error for reflection normalization.

11.4 Storage Instructions

Before first use store your Analyzer in the factory package at environment temperature from 0 to +40 °C and relative humidity up to 80% (at 25 °C).

After you have removed the factory package store the Analyzer at environment temperature from +10 to +35 °C and relative humidity up to 80% (at 25 °C).

Ensure to keep the storage facilities free from dust, fumes of acids and alkalies, aggressive gases, and other chemicals, which can cause corrosion.

Appendix 1

Default values defined in the process of the initial factory setup.

Parameter Description	Default Setting	Parameter Setting Object
Touchstone Data Format	RI - Real-Imaginary	Analyzer
Allocation of Channels	1	Analyzer
Active Channel Number	1	Analyzer
Marker Value Identification Capacity (Stimulus)	8 digits	Analyzer
Marker Value Identification Capacity (Response)	5 digits	Analyzer
Vertical Divisions	10	Channel
Channel Title Bar	OFF	Channel
Channel Title	Empty	Channel
Traces per Channel	1	Channel
Active Trace Number	1	Channel
Sweep Type	Linear	Channel
Number of Sweep Points	201	Channel
Stimulus Start Frequency	Instrument min.	Channel
Stimulus Stop Frequency	Instrument max.	Channel
Stimulus Power Level	High	Analyzer
Stimulus IF Bandwidth	10 kHz	Channel
Sweep Measurement Delay	0 sec.	Channel
Sweep Range Setting	Start / Stop	Channel
Number of Segments	1	Channel
Points per Segment	2	Channel

Parameter Description	Default Setting	Parameter Setting Object
Segment Start Frequency	Instrument min.	Channel
Segment Stop Frequency	Instrument min.	Channel
Segment Sweep IF Bandwidth	10 kHz	Channel
Segment Sweep Delay (Table Display)	OFF	Channel
Segment Sweep IFBW (Table Display)	OFF	Channel
Trigger Mode	Continuous	Analyzer
Table of Calibration Coefficients	Empty	Channel
Error Correction	ON	Analyzer
Trace Scale	10 dB/division	Trace
Reference Level Value	0 dB	Trace
Reference Level Position	5 Div.	Trace
Phase Offset	0°	Trace
Electrical Delay	0 sec.	Trace
Trace Display Format	Return Loss (dB)	Trace
Start Distance	-1.49 m	Trace
Stop Distance	1.49 m	Trace
Time Domain Kaiser Window	Normal	Channel
Number of Markers	0	Trace