R&S®RTO2000 Oscilloscope User Manual

1332972502
Version 19

ROHDE&SCHWARZ
Make ideas real

This manual describes the following R&S®RTO models with firmware version 5.10:

- R&S®RTO2002 (1329.7002K02)
- R&S®RTO2004 (1329.7002K04)
- R&S®RTO2012 (1329.7002K12)
- R&S®RTO2014 (1329.7002K14)
- R&S®RTO2022 (1329.7002K22)
- R&S®RTO2024 (1329.7002K24)
- R&S®RTO2032 (1329.7002K32)
- R&S®RTO2034 (1329.7002K34)
- R&S®RTO2044 (1329.7002K44)
- R&S®RTO2064 (1329.7002K64)

Mühldorfstr. 15, 81671 München, Germany
Phone: +49 89 41 29 - 0
Email: info@rohde-schwarz.com
Internet: www.rohde-schwarz.com

1332.9725.02 | Version 19 | R&S®RTO2000

Throughout this manual, products from Rohde & Schwarz are indicated without the ® symbol, e.g. R&S®RTO is indicated as R&S RTO.

Contents

1 Safety and regulatory information

The product documentation helps you to use the product safely and efficiently. Follow the instructions provided here and in the Chapter 1.1, "Safety instructions", on page 11.

Intended use

The R&S RTO oscilloscope is designed for measurements on circuits that are only indirectly connected to the mains or not connected at all. It is not rated for any measurement category.

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

Where do I find safety information?

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

- In Chapter 1.1, "Safety instructions", on page 11. The same information is provided in many languages as printed "Safety Instructions". The printed "Safety Instructions" are delivered with the product.
- Throughout the documentation, safety instructions are provided when you need to take care during setup or operation.

1.1 Safety instructions

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

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

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

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

In these safety instructions, the term "product" covers instruments (oscilloscopes), probes and their accessories.

Lifting and carrying the instrument

Check the data sheet for the maximum weight of the instrument. A single person can only carry a maximum of 18 kg safely depending on age, gender and physical condition. If your instrument is heavier than 18 kg, do not move or carry it by yourself.

Use the instrument handles to move or carry the instrument. Do not use the mounted accessories instead of the handles. Accessories are not designed to carry the weight of the instrument.

To move the instrument safely, you can use lifting or transporting equipment such as lift trucks and forklifts. Follow the instructions provided by the equipment manufacturer.

Choosing the operating site

Only use the product indoors. The product casing is not waterproof. Water that enters can electrically connect the casing with live parts, which can lead to electric shock, serious personal injury or death if you touch the casing. If Rohde & Schwarz provides accessories designed for your product, e.g. a carrying bag, you can use the product outdoors.

You can operate the product up to the altitude specified in the data sheet. The lowest specified altitude for a product of the measurement setup defines the altitude for the complete setup.

The product is suitable for pollution degree 2 environments where nonconductive contamination can occur. For more information on environmental conditions such as ambient temperature and humidity, see the data sheet.

Setting up the product

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

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

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

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

Connecting to power and grounding

The mains power supply input of the instrument complies with overvoltage category II. It has to be connected to a fixed installation used to supply energy-consuming equipment such as household appliances and similar loads. Be aware that electrically powered products have risks, such as electric shock, fire, personal injury or even death.

Take the following measures for your safety:

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

Performing measurements

Take the following measures for your safety:

- To ascertain voltage-free state, use an appropriate voltage tester. Any measurement setup including an oscilloscope is not suitable for this purpose.
- The maximum input voltage on channel inputs and the external trigger input must not exceed the value specified in the data sheet.
- Observe all voltage and current ratings of the instrument, the probes, and the accessories. Limits and ratings are marked on the products and listed in the data sheets.
  Consider that the rated voltage depends on the frequency. The voltage limitation curves or values are provided in the data sheet. Do not exceed the maximum measurement voltage from the probe tip to the probe reference lead.
- Never cause any short circuits when measuring sources with high output currents.
- Use only probes and accessories that comply with the measurement category (CAT) of your measurement task. The measurement category of the products is defined in the data sheet. If you use other than Rohde & Schwarz accessories, make sure that they are suitable for the instrument and the measurement task.
- Set the correct attenuation factor on the instrument according to the probe being used. Otherwise, the measurement results do not reflect the actual voltage level, and you might misjudge the actual risk.

- When working with high voltages and current probes, observe the additional operating conditions specified in this safety instructions.
- The probe pins are extremely pointed and can easily penetrate clothes and the skin. Handle the probe pins with great care. To exchange a probe pin, use tweezers or pliers to avoid injuries. When transporting the accessories, always use the box supplied with the probe.
- Prevent the probe from receiving mechanical shock. Avoid putting excessive strain on the probe cable or exposing it to sharp bends. Touching a broken cable during measurements can cause injuries.
- Set up all probe connections to the instrument before applying power.

Working with hazardous voltages

Voltages higher than 30 V RMS, or 42 V peak, or 60 V DC are regarded as hazardous contact voltages. Direct contact with them can cause serious injuries.

Make sure that only electrically skilled persons use the products for measurements on hazardous contact voltages. These working conditions require special education and experience to perceive risks and to avoid hazards which electricity can create.

When working with hazardous contact voltages, use protective measures to preclude direct contact with the measurement setup:

- Do not touch exposed connections and components when power is applied.
- Switch off the test circuit while connecting and disconnecting probe leads.
- Use only insulated voltage probes, test leads and adapters.
- Make sure that the input leads fulfill the safety requirements for your measurement. The delivered input leads might have a jacket wear indicator that indicates a worn jacket by different jacket color. In this case, do not use the input lead. Replace it with a new one.
- Do not use 4 mm banana plugs without protection against contact.

Working with current probes

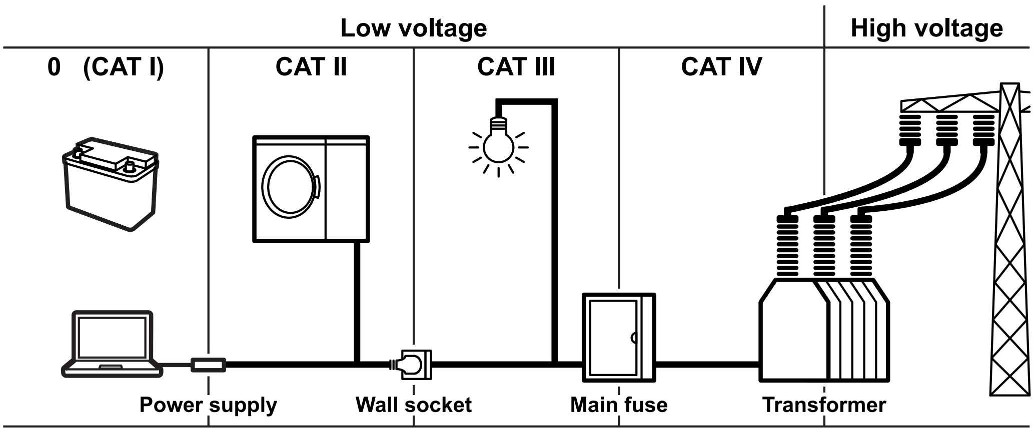
When working with current probes, you can measure high-frequency currents or currents that contain high-frequency components.

- Switch off the test circuit while connecting the probe.
- Do not attach the clamp to bare unisolated conductors. To avoid injury from a short circuit, measure at a location on an insulated wire where the insulation is sufficient for the circuit voltage.
- Connect the probe only to the secondary side of a breaker. With this measure, you avoid injury, if a short circuit occurs.
- The following effects can cause burns and fire or damage to the measurement site:
  - Eddy current loss can cause heating of the sensor head.
  - Dielectric heating can cause heating of cord insulation and other materials.

Measurement categories

IEC 61010-2-030 defines measurement categories that rate instruments on their ability to resist short transient overvoltages that occur in addition to the working voltage. Use the measurement setup only in electrical environments for which they are rated.

- 0 - Instruments without rated measurement category
  For measurements performed on circuits not directly connected to mains, for example, electronics, circuits powered by batteries, and specially protected secondary circuits. This measurement category is also known as CAT I.
- CAT II:
  For measurements performed on circuits directly connected to the low-voltage installation by a standard socket outlet, for example, household appliances and portable tools.
- CAT III:
  For measurements performed in the building installation, such as junction boxes, circuit breakers, distribution boards, and equipment with permanent connection to the fixed installation.
- CAT IV:
  For measurements performed at the source of the low-voltage installation, such as electricity meters and primary overcurrent protection devices.



Cleaning the product

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

Meaning of safety labels

Safety labels on the product warn against potential hazards.

	Potential hazard Read the product documentation to avoid personal injury or product damage.
	Electrical hazard Indicates live parts. Risk of electric shock, fire, personal injury or even death.
	Protective conductor terminal Connect this terminal to a grounded external conductor or to protective ground. This protects you against electric shock should an electric problem occur.

1.2 Labels on the product

Labels on the casing inform about:

- Personal safety, see "Meaning of safety labels" on page 15
- Product and environment safety, see Table 1-1
- Identification of the product

Table 1-1: Labels regarding product and environment safety

	Chassis grounding terminal
	Take care when handling electrostatic sensitive devices.
	Labeling in line with EN 50419 for disposal of electrical and electronic equipment after the product has come to the end of its service life. For more information, see "Disposing electrical and electronic equipment" on page 2582.

1.3 Warning messages in the documentation

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

WARNING

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

CAUTION

Potentially hazardous situation. Could result in minor or moderate injury if not avoided.

NOTICE

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

1.4 Korea certification class A

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

2 Preface

2.1 Key features

The R&S RTO oscilloscopebrings various benefits in your daily work. Outstanding key features are:

Best oscilloscope performance:

- Up to 16-bit vertical resolution
- Trigger on any detail you can see
- Quickly find signal faults with 1 Million waveforms/s
- Integrated spectrum analysis

Widest range of capabilities:

- Fast acquisition rate even with industry leading 2 Gsample deep memory
- First zone trigger in time and frequency domain
- Analyze previous acquisitions - always available in history buffer
- Deep toolset for signal analysis

Powerful user interface:

- High-resolution 12.1” capacitive touchscreen with gesture support
- Easily customizable waveform display with SmartGrid
- Fast access to important tools
- Documentation at the press of a button

Engineered for multi-domain challenges:

- Class leading MSO
- Trigger and decode for serial protocols
- Advanced spectrum analysis
- Integrated arbitrary waveform and pattern generator
- Power measurements
- Signal integrity testing with advanced eye and jitter analysis
- Automatic compliance tests
- EMI debugging with oscilloscopes
- IQ and RF analysis
- Extensive probe portfolio for voltage, current and more

For a detailed specification refer to the data sheet.

2.2 Documentation overview

This section provides an overview of the R&S RTO user documentation.

2.2.1 Manuals and instrument help

You find the manuals on the product page at:

www.rohde-schwarz.com/manual/rto

Getting started manual

Introduces the R&S RTO and describes how to set up and start working with the instrument, and describes basic operations. A printed English version is included in the delivery. Editions in other languages are available on the product website.

Instrument help

The help offers quick, context-sensitive access to the complete information for the firmware basic functionality and applications.

User manual (Instrument)

Describes all instrument functions in detail. It also provides an introduction to remote control, a complete description of the remote control commands with programming examples, and information on maintenance and instrument interfaces. Includes the contents of the getting started manual.

The *online version* of the user manual provides the complete contents for immediate display on the internet.

Manuals for compliance test options

For compliance test options, extra test procedure manuals are available. Test fixtures are described in printed manuals, which are delivered with the fixture.

- The following test procedure manuals are available:
  – USB 2.0 Compliance Test Procedures
  – Ethernet Compliance Tests Procedures
  – MIPI D-PHY Compliance Tests Procedures
  – eMMC Compliance Tests Procedures
  – PCIe Compliance Tests Procedures
  – DDR3 Compliance Tests Procedures
  – ScopeSuite Automation
- The following test fixture manuals are available:
  – R&S RT-ZF1 USB 2.0 Compliance Test Fixture Set
  – R&S RT-ZF2 Ethernet Compliance Test Fixture Set
  – R&S RT-ZF3 Frequency Converter Board (100BASE-T1)
  – R&S RT-ZF4 10BASE-Te Test Fixture

– R&S RT-ZF5 Ethernet Probing Fixture
– R&S RT-ZF6 Frequency Converter Board (1000BASE-T1)
– R&S RT-ZF7 Automotive Ethernet T&D Fixture
– R&S RT-ZF7A SMA Adaptor
– R&S RT-ZF8 Automotive Ethernet Compliance Test Fixture

Safety instructions

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

Instrument security procedures manual

Deals with security issues when working with the R&S RTO in secure areas.

Service Manual

Describes the performance test for checking the rated specifications, module replacement, firmware update, troubleshooting and fault elimination, and contains mechanical drawings and spare part lists. The service manual is available for registered users on the global Rohde & Schwarz information system (GLORIS, https://gloris.rohde-schwarz.com).

2.2.2 Data sheet and brochure

The data sheet contains the technical specifications of the R&S RTO. It also lists the options with their order numbers and optional accessories. The brochure provides an overview of the instrument and deals with the specific characteristics.

See www.rohde-schwarz.com/brochure-datasheet/rto

2.2.3 Release notes, open source acknowledgment

The release notes list new features, improvements and known issues of the current firmware version, and describe the firmware installation. The open source acknowledgment document provides verbatim license texts of the used open source software. It can also be read directly on the instrument.

See www.rohde-schwarz.com/firmware/rto.

2.2.4 Application notes, application cards, videos

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

See www.rohde-schwarz.com/application/rto and Oscilloscopes Application Videos - Media Center

2.3 Options Described in this Document

In addition to the base unit, the following options are described in this documentation:

Type	Designation	Order No.
R&S RTO-B1	MSO	1304.9901.03
R&S RTO-B6	Waveform and pattern generator	1329.7054.02
R&S RTO-B7	Pulse source	1333.2001.02
R&S RTO-K1	I²C and SPI serial triggering and decoding	1329.7260.02
R&S RTO-K2	UART/RS-232/RS-422/RS-485 serial triggering and decoding	1329.7277.02
R&S RTO-K3	CAN and LIN serial triggering and decoding	1329.7283.02
R&S RTO-K4	FlexRay™ serial triggering and decoding	1329.7290.02
R&S RTO-K5	I²S (audio) serial triggering and decoding	1329.7302.02
R&S RTO-K6	MIL-STD-1553 serial triggering and decoding	1329.7319.02
R&S RTO-K7	ARINC 429 serial triggering and decoding	1329.7325.02
R&S RTO-K8	Ethernet serial triggering and decoding	1329.7331.02
R&S RTO-K9	CAN-FD serial triggering and decoding	1329.7348.02
R&S RTO-K10	SENT serial triggering and decoding	1329.7354.02
R&S RTO-K11	I/Q software interface	1329.7360.02
R&S RTO-K12	Jitter analysis	1329.7377.02
R&S RTO-K13	Clock data recovery	1329.7383.02
R&S RTO-K133	Advanced jitter analysis	1801.4832.02
R&S RTO-K134	Advanced noise analysis	1802.9450.02
R&S RTO-K18	Spectrum Analysis	1329.7425.02
R&S RTO-K19	Zone trigger	1329.7431.02
R&S RTO-K31	Power analysis	1329.7502.02
R&S RTO-K35	Bus analysis for I2C, SPI, RS232/UART, CAN, CAN-FD, LIN, SENT, 100BASE-Tx and 100BASE-T1. Requires corresponding protocol decoding option.	1801.2846.02
R&S RTO-K40	MIPI RFFE serial triggering and decoding	1329.7519.02
R&S RTO-K42	MIPI D-PHY serial triggering and decoding	1329.7525.02
R&S RTO-K44	MIPI M-PHY serial triggering and decoding	1333.0267.02
R&S RTO-K50	Custom Manchester and NRZ serial triggering and decoding	1329.7531.02
R&S RTO-K52	8b10b serial triggering and decoding	1329.7548.02
R&S RTO-K55	MDIO serial triggering and decoding	1329.7554.02

Type	Designation	Order No.
R&S RTO-57	100BASE-T1 serial triggering and decoding	1333.0596.02
R&S RTO-K58	1000BASE-T1 serial triggering and decoding	1801.4503.02
R&S RTO-K60	USB 1.0/1.1/2.0/HSIC serial triggering and decoding	1329.7560.02
R&S RTO-K61	USB 3.1 Gen 1 serial triggering and decoding	1326.3112.02
R&S RTO-K63	USB-PD serial triggering and decoding	1326.3135.02
R&S RTO-K64	USB SSIC serial triggering and decoding	1337.9123.02
R&S RTO-K65	SpaceWire serial triggering and decoding	1326.2868.02
R&S RTO-K72	PCI Express 1.x/2.x serial triggering and decoding	1326.3741.02
R&S RTO-K76	CXPI serial triggering and decoding	1326.3170.02
R&S RTO-K91	DDR3/DDR3L/LPDDR3 debug & compliance test. Decode and debug is described in this user manual. The compliance test is described in a separate manual.	1337.8891.02
R&S RTO-K121	Deembedding base option	1326.3058.02
R&S RTO-K130	TDR/TDT analysis (requires pulse source option R&S RTO-B7) For R&S RTO with at least 2 GHz bandwidth	1326.3087.02

2.4 Conventions used in the documentation

2.4.1 Typographical conventions

The following text markers are used throughout this documentation:

Convention	Description
"Graphical user interface elements"	All names of graphical user interface elements on the screen, such as dialog boxes, menus, options, buttons, and softkeys are enclosed by quotation marks.
[Keys]	Key and knob names are enclosed by square brackets.
`Filenames, commands, program code`	Filenames, commands, coding samples and screen output are distinguished by their font.
Input	Input to be entered by the user is displayed in italics.
Links	Links that you can click are displayed in blue font.
"References"	References to other parts of the documentation are enclosed by quotation marks.

2.4.2 Conventions for procedure descriptions

When operating the instrument, several alternative methods may be available to perform the same task. In this case, the procedure using the touchscreen is described. Any elements that can be activated by touching can also be clicked using an additionally connected mouse. The alternative procedure using the keys on the instrument or the on-screen keyboard is only described if it deviates from the standard operating procedures.

The term "select" may refer to any of the described methods, i.e. using a finger on the touchscreen, a mouse pointer in the display, or a key on the instrument or on a keyboard.

3 Getting started

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

3.1 Preparing for use

Here, you can find basic information about setting up the instrument for the first time or when changing the operating site.

3.1.1 Lifting and carrying

See: "Lifting and carrying the instrument" on page 12.

3.1.2 Unpacking and checking

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

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

3.1.3 Choosing the operating site

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

See also "Choosing the operating site" on page 12.

Electromagnetic compatibility classes

The electromagnetic compatibility (EMC) class indicates where you can operate the product. The EMC class of the product is given in the data sheet under "General data".

- Class B equipment is suitable for use in:
  – Residential environments

- Environments that are directly connected to a low-voltage supply network that supplies residential buildings

- Class A equipment is intended for use in industrial environments. It can cause radio disturbances in residential environments due to possible conducted and radiated disturbances. It is therefore not suitable for class B environments.
  If class A equipment causes radio disturbances, take appropriate measures to eliminate them.

3.1.4 Setting up the product

When setting up the instrument, follow the safety instructions:

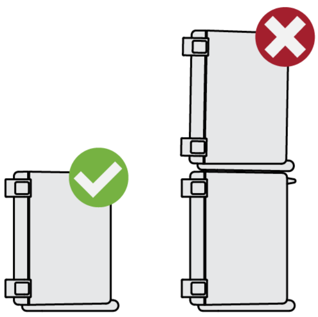
- "Setting up the product" on page 12
- "Intended use" on page 11

3.1.4.1 Placing the product on a bench top

For standalone operation, place the instrument on a horizontal bench with even, flat surface. The instrument can be used in horizontal position, standing on its feet, or with the support feet on the bottom extended.

To place the product on a bench top

1. Place the product on a stable, flat and level surface. Ensure that the surface can support the weight of the product. For information on the weight, see the data sheet.
2. **CAUTION!** Foldable feet can collapse. See "Setting up the product" on page 12.
   Always fold the feet completely in or out. With folded-out feet, do not place anything on top or underneath.
3. **CAUTION!** The product can fall over and cause injury. The top surface is too small for stacking. Never stack another product on top of the product.

   As an alternative, you can mount several products in a rack.
4. **NOTICE!** Overheating can damage the product.
   Prevent overheating as follows:

- Keep a minimum distance of 10 cm between the fan openings of the product and any object in the vicinity.
- Do not place the product next to heat-generating equipment such as radiators or other products.

3.1.4.2 Mounting the product in a rack

The instrument can be installed in a rack using a rack adapter kit. The order number is given in the data sheet. The installation instructions are part of the adapter kit.

To prepare the rack

1. Observe the requirements and instructions in "Setting up the product" on page 12.
2. **NOTICE!** Insufficient airflow can cause overheating and damage the product.
   Design and implement an efficient ventilation concept for the rack.

To mount the R&S RTO in a rack

1. Use an adapter kit that fits the dimensions of the R&S RTO to prepare the instrument for rack mounting. For information on the dimensions, see data sheet.
   a) Order the rack adapter kit designed for the R&S RTO. For the order number, see data sheet.
   b) Mount the adapter kit. Follow the assembly instructions provided with the adapter kit.
2. Push the product onto the shelf until the rack brackets fit closely to the rack.
3. Tighten all screws at the rack brackets with a tightening torque of 1.2 Nm to secure the product at the rack.

To unmount the product from a rack

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

3.1.5 Considerations for test setup

Observe safety instructions, see "Performing measurements" on page 13.

Cable selection and electromagnetic interference (EMI)

Electromagnetic interference (EMI) can affect the measurement results.

To suppress electromagnetic radiation during operation:

- Use high-quality shielded cables, for example, double-shielded RF and LAN cables.

- Always terminate open cable ends.
- Ensure that connected external devices comply with EMC regulations.

Measuring accessories

Use only probes and measuring accessories that comply with IEC 61010-031.

Signal input and output levels

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

Preventing electrostatic discharge (ESD)

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

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

Ground yourself to prevent electrostatic discharge damage:

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

During operation, if the firmware observes a serious unexpected disturbance (e.g. due to ESD), it resets some hardware components and initiates a new alignment to ensure proper instrument functioning. Then it restores the user settings to the state before the disturbance.

3.1.6 Connecting to power

For safety information, see "Connecting to power and grounding" on page 12.

The R&S RTO can be used with different AC power voltages and adapts itself automatically to it.

The nominal ranges are:

- 100 V to 240 V AC at 50 Hz to 60 Hz and 400 Hz, with maximal 10% voltage fluctuation on line
- max. 5.5 A to 2.3 A

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

   The required ratings are listed next to the AC power connector and in the data sheet.

3.1.7 Switching on or off

The instrument is switched on or off with the power switch and the [Power] key. The [Power] key is located in the bottom left corner of the front panel. The power switch is located at the rear panel of the instrument.

Table 3-1: Overview of power states

Status	LED	Power switch
Off	(unlighted)	[0] (off)
Standby	orange	[I] (on)
Ready	green	[I] (on)

To switch on the product

The product is off but connected to power.

1. Set the switch on the power supply to position [I].

   The power key on the front panel lights up.

2. Press the [Power] key on the front panel.

   The instrument performs a system check, boots the operating system, and then starts the R&S RTO firmware.
   The [Power] key turns green and the illuminated keys on the front panel light up. If the previous session was terminated regularly, the oscilloscope uses the last settings.

Before you start measurements, be sure to comply with the warm-up phase specified in the data sheet.

To shut down the product

The product is in the ready state.

► Press the [Power] key.

   All current settings are saved, and the software shuts down. The [Power] key turns orange. The standby power supplies only the power switch circuits.

The "Exit" icon in the "Menu" shuts down only the firmware application. To shut down the instrument completely, use the [Power] key.

To disconnect from power

The product is in the standby state.

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

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

   The LED of the [Power] key is switched off.

2. Disconnect the product from the power source.

3.1.8 Connecting external devices

The following interfaces for external devices are provided:

- USB connectors at the front and rear panel of the instrument
- Monitor connectors DVI-D and DisplayPort at the rear panel of the instrument

3.1.8.1 Connecting USB devices

The USB interfaces on the front and rear panels allow you to connect USB devices directly to the instrument. The number of USB connectors can be increased by using USB hubs. Due to the large number of available USB devices, there is almost no limit to the expansions that are possible with the R&S RTO.

The following USB devices can be useful, for example:

- USB flash drives to save screenshots and measurement results, and for easy installation of firmware applications
- Keyboard and/or mouse to simplify the operation and the entry of data, comments, filenames, etc.

You can connect or disconnect all USB devices during operation of the instrument.

Installing USB devices on R&S RTO is easy under the Windows operating system, because all USB devices are plug&play. After a device is connected to the USB interface, Windows automatically searches for a suitable device driver.

If the operating system does not find a suitable driver, it prompts you to specify a directory that contains the driver software. If the driver software is on a storage media, connect the appropriate drive to the instrument before proceeding. If the instrument is integrated in a network, you can also install driver data stored in a network directory.

When a USB device is disconnected from the R&S RTO, Windows immediately detects the change in hardware configuration and deactivates the corresponding driver.

The properties of external USB devices are configured in the operating system, not in the R&S RTO software. It is recommended that you use mouse and keyboard to access and modify the settings of the Windows operating system.

To access Windows, press the Windows key on the external keyboard, or select "Menu" > "Minimize Application" on the R&S RTO menu.

Connecting a USB flash drive

If the installation of a USB flash drive is successful, Windows informs you that the device is ready to use. The device is made available as a new drive ("D:") and is displayed in Windows Explorer. The name of the drive depends on the manufacturer.

Connecting a keyboard

The keyboard is detected automatically when it is connected. The default input language is English – US.

To configure the keyboard properties:

1. Tap the "Find" icon (magnifier) on the Windows taskbar.
2. Type *keybord*.
3. Select "Edit language and keyboard options".

Connecting a mouse

The mouse is detected automatically when it is connected. To configure the mouse properties:

1. Tap the "Find" icon (magnifier) on the Windows taskbar.
2. Type *mouse*.
3. Select "Mouse settings".

3.1.8.2 Connecting an external monitor

You can connect an external monitor or projector to the R&S RTO. The following connectors are available:

- "DVI-D" on page 34
- "DisplayPort" on page 34

Before connecting an external monitor, ensure that the monitor and the R&S RTO are connected to a ground contact. Otherwise the instrument can be damaged.

After connecting an additional monitor or projector to the instrument, configure it for usage. The relevant settings are Windows settings but you can configure the displays directly in the instrument setup.

1. Check the input type of the monitor or projector. Make sure to select the correct cable. To use a VGA monitor, you need an active DVI-D to VGA adapter.
2. Open "Menu" > "Settings" > "Display".
3. Select the "Monitors" tab.
4. Select how to display the screen.
5. To access Windows display settings, tap "Additional settings".

The touchscreen of the R&S RTO has a screen resolution of 1280 x 800 pixel. Many external monitors have a higher screen resolution. If the screen resolution of the monitor is set higher than the instrument's resolution, the application window uses a 1280 x 800 area of the monitor display. For full screen display, adjust the monitor's screen resolution using "Additional display settings".

3.2 Instrument tour

This chapter describes the front and rear panels of the instrument including all function keys and connectors.

3.2.1 Front panel

The front panel of the R&S RTO is shown in Figure 3-1. The function keys are grouped in functional blocks to the left and the right of the touchscreen. Below the screen, various connectors are located.

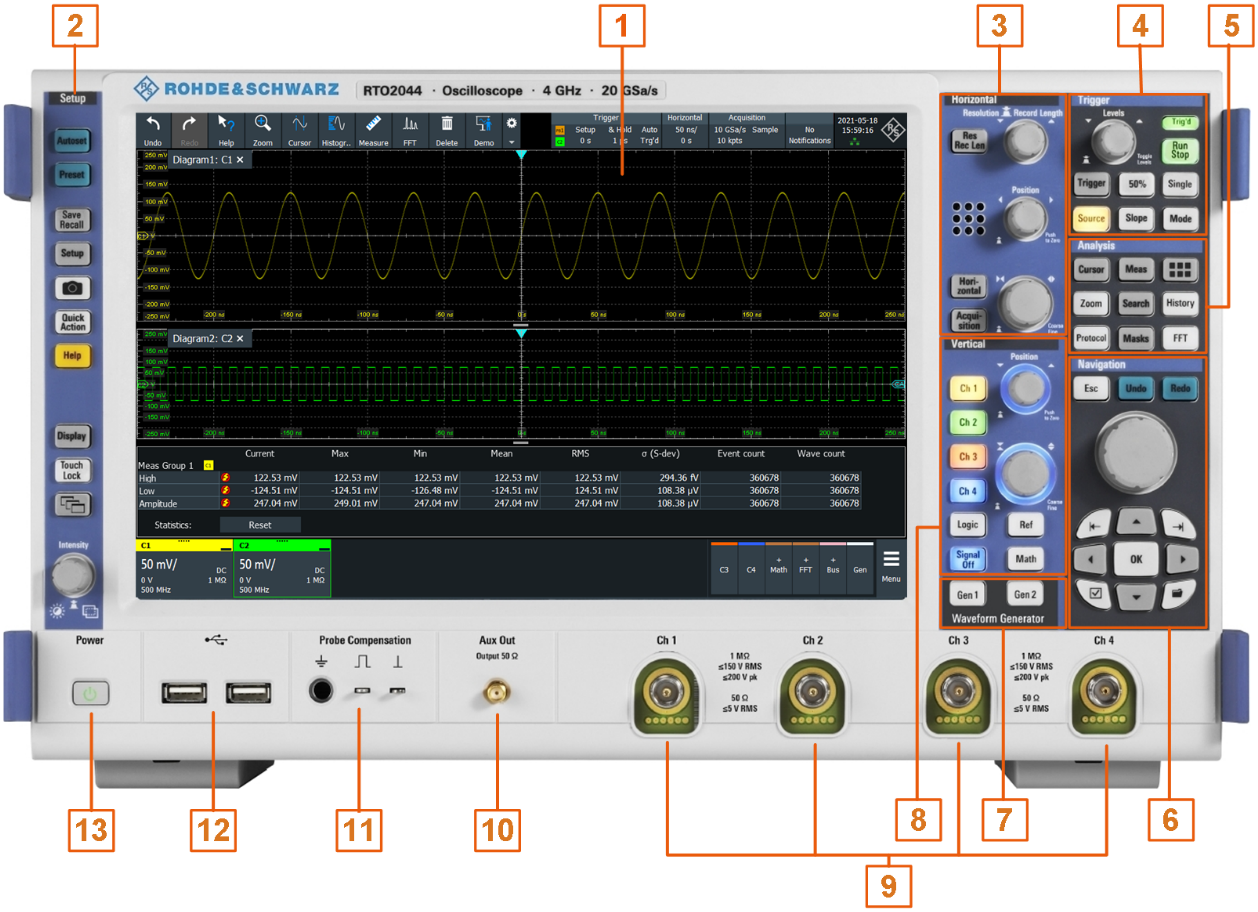


Figure 3-1: Front panel of R&S RTO2044 with 4 input channels

1 = Touchscreen
2 = [Setup] controls
3 = [Horizontal] controls
4 = [Trigger] controls
5 = [Analysis] keys
6 = [Navigation] controls
7 = [Waveform Generator] keys
8 = [Vertical] controls
9 = Input channels
10 = [Aux Out] connector

11 = Connectors for probe compensation and grounding
12 = USB connectors
13 = [Power] key

3.2.1.1 Input connectors

The R&S RTO has two or four channel inputs to connect the input signals using active and passive probes.

The input connectors are provided with a special Rohde & Schwarz active probe interface, and they are BNC compatible. Thus, the instrument can automatically detect passive probes with standard BNC connector and active Rohde & Schwarz probes having the Rohde & Schwarz probe interface.

The input impedance is selectable, the values are 50 Ω and 1 MΩ.

The maximum input voltage is 200 V peak, 150 V RMS at 1 MΩ input impedance and 5 V RMS at 50 Ω input impedance.

3.2.1.2 Other front panel connectors

Besides the input connectors, the instrument has USB connectors and probe compensation connectors at the front panel.

[USB]
Two USB type A connectors that comply with standard USB 2.0. They are used to connect devices like keyboard, mouse and USB flash drive.

Note: Electromagnetic interference (EMI) can affect the measurement results. To avoid any impact, do not use USB connecting cables exceeding 1 m.

Probe Compensation
Probe compensation terminal to support adjustment of passive probes to the oscilloscope channel.

⏚	Protective earth conductor, 4 mm banana jack to connect ground of DUT and test fixtures, and wrist strap.
⎍	Square wave signal for probe compensation, 1 kHz and 1 V_{pp}.
⊥	Ground connector for probes.

[Aux Out]
Output of the internal calibration signal, if the signal is configured to external destination.

3.2.2 Rear panel

Figure 3-2 shows the rear panel of the R&S RTO.

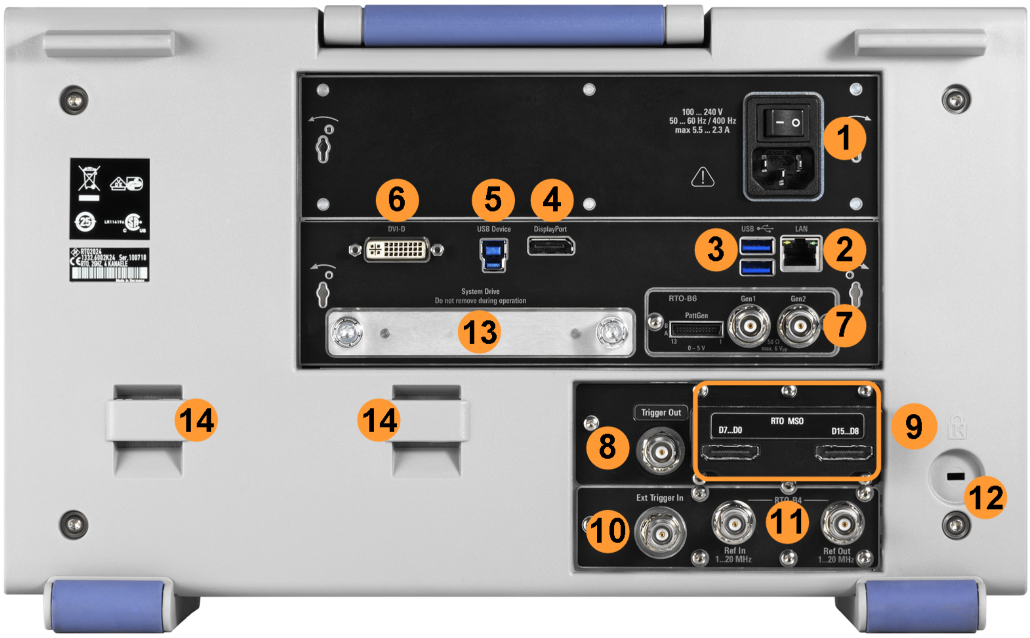


Figure 3-2: Rear panel of R&S RTO

1 = AC power supply connector and main power switch
2 = LAN connector
3 = USB connectors, type A
4 = DisplayPort connector
5 = USB Device connector, type B
6 = DVI-D connector for external monitor
7 = Option slot for R&S RTO-B6 (waveform generator, shown in figure) or R&S RTO-B7 (pulse source)
8 = External trigger output
9 = Option slot for R&S RTO-B1 (MSO, shown in figure) or R&S RTO-B1E (for R&S RT-ZVC) , or R&S RTO-B10 (GPIB)
10 = External trigger input
11 = Optional OCXO with input and output of the reference signal (option R&S RTO-B4)
12 = Kensington lock slot to secure the instrument against theft
13 = Optional exchangeable hard disk: solid state disk (option R&S RTO-B18) or standard hard disk drive (option R&S RTO--B19)
14 = Lugs to attach the accessory bag

AC power supply connector and main power switch

Connection to the AC power line. The R&S RTO can be used with different AC power voltages and adapts itself automatically to it. The nominal voltage and frequency ranges are displayed on the rear panel and quoted in the data sheet.

If grounding is *not* ensured by the mains system, ground the oscilloscope using the protective earth conductor on the front panel and an appropriate cable.

The AC main power switch also interrupts the power supply of the OCXO (option OCXO Reference Frequency, R&S RTO-B4).

When you power up the instrument, be sure to comply with the warm-up phase specified in the data sheet before you start measurements.

See also: Chapter 3.1.6, "Connecting to power", on page 27.

USB
Two USB type A connectors that comply with standard USB 3.1 gen 1. They are used to connect devices like keyboard, mouse and flash drive to store and reload instrument settings and measurement data.

Note: Electromagnetic interference (EMI) can affect the measurement results. To avoid any impact, do not use USB connecting cables exceeding 1 m.

LAN
8-pin RJ-45 connector used to connect the instrument to a Local Area Network (LAN). It supports up to 1000 Mbit/s (10/100/1000BASE-T Ethernet).

USB Device
USB 3.0 interface of type B (device USB), to be used for remote control of the instrument.

DisplayPort
DisplayPort connector for an external monitor or projector. It supports DisplayPort version 1.1a.

DVI-D
Digital connector for an external monitor or projector. The monitor shows the complete content of the instrument's screen.

See also: Chapter 3.1.8.2, "Connecting an external monitor", on page 30.

Ext Trigger In
The BNC connector for external trigger input is used to control the measurement by an external signal. The input impedance can be selected in the trigger configuration, the values are 50 Ω and 1 MΩ. The trigger level can be set from -5 V to 5 V. The maximum input voltage is 30 V RMS at 1 MΩ input impedance and 7 V RMS at 50 Ω input impedance.

Trigger Out
The BNC connector for external trigger output is used to provide the internal trigger signal of the oscilloscope to trigger other instruments for synchronized measurements.

When a trigger occurs, the R&S RTO creates a pulse of 5 V with a source impedance of 50 Ω and delivers it to the external trigger output. The instrument can also send the pulse on mask test violation or violation of measurement limits and margins.

If the connector is terminated with 50 Ω, the signal level is 2.5 V (50 mA). With 1 MΩ termination, the level is 5 V. A short-circuit of the connector to ground creates current of 100 mA.

To enable the trigger out signal, select "Trigger" menu > "Ctrl/Action". Here you also adjust polarity, delay, and length of the pulse. The default is a positive pulse of 100 ns. The minimum delay is 800 ns.

Mixed signal option R&S RTO-B1 / digital extension port R&S RTO-B1E

The mixed signal option R&S RTO-B1 provides logic analizer functionality and 16 digital channels. It also can connect the R&S RT-ZVC multi-channel power probe. The connectors are used to connect two logical probes with 8 digital channels each (D0 to D7 and D8 to D15), or two flat interface cables of R&S RT-ZVC.

The digital extension port R&S RTO-B1E provides the connectors for the R&S RT-ZVC multi-channel power probe without MSO. The connectors are used to connect two flat interface cables of R&S RT-ZVC.

One of the options can be installed in the option slot at the rear panel.

The maximum input voltage is 40 V peak at 100 kΩ input impedance. The maximum input frequency for a signal with the minimum input voltage swing of 500 mV (V_{pp}) is 400 MHz. For detailed specifications, refer to the data sheet.

OCXO option R&S RTO-B4

Optional [Ref In] (left) and [Ref Out] (right) connectors coming with option R&S RTO-B4 OCXO 10 MHz.

The input frequency ranges from 1 MHz to 20 MHz in 1 MHz steps. The input impedance is 50 Ω.

The output frequency of the OCXO is 10 MHz, the impedance is 50 Ω. For detailed specifications, refer to the data sheet.

Waveform generator option R&S RTO-B6

The waveform generator generates various function and arbitrary waveforms, sweeps, and parallel patterns. For detailed specifications, refer to the data sheet.

The option can be installed in the option slot at the rear panel.

The module provides the following connectors:

[Gen1, Gen2]	BNC connectors
[PattGen]	Connector for the pattern generator

Pulse source option R&S RTO-B7

The pulse source outputs a symmetrical differential pulse signal. For detailed specifications, refer to the data sheet.

The option can be installed in the option slot at the rear panel.

The module has four connectors.

- Out, $\overline{\text{Out}}$: 2.92 mm connectors (K type) for pulse signal output
- Ref, $\overline{\text{Ref}}$: 2.92 mm connectors (K type) for reference signal output

GBIP option R&S RTO-B10

The GBIP option provides a GBIP interface and connector for remote control. For detailed specifications, refer to the data sheet.

3.2.3 Keys and controls

3.2.3.1 Power key

The [Power] key is located on the lower left corner of the front panel. It starts up and shuts down the instrument's software.

The light of the key shows the instrument state:

- Standby, the main power switch is on, the software is shut down: orange.
- The instrument is ready for operation: green.

See also: Chapter 3.1.7, "Switching on or off", on page 28.

3.2.3.2 Setup controls

Setup keys set the instrument to a defined state, change basic settings, and provide saving and help functions. The intensity rotary knob adjusts the display contrast for several display elements.

[Autoset]
The instrument analyzes the enabled channel signals, and adjusts appropriate horizontal, vertical, and trigger settings to display stable waveforms.

[Preset]
Resets the instrument to a default state. All measurements, mask tests, zoom, and most individual settings are deleted, and all channels except for channel 1 are disabled. You can define preset configurations and save them to a file. The [Preset] key can be configured to set either factory defaults or a user-defined preset configuration.

[Save Recall]
Opens and closes the "File" dialog box, where you can:

- Save instrument settings (user settings)
- Load instrument settings which were saved before
- Save waveform data and measurement results
- Define a naming pattern for autonaming of files

[Setup]
Opens and closes the "Setup" dialog box, where you can:

- Access Windows configuration and install firmware updates
- Configure the touchscreen
- Check and install option keys for software options
- Check availability of hardware options
- Configure remote settings, LAN settings, and GPIB

Camera
Starts a saving action, or opens a report. By default, the key saves a screenshot of the waveform display. You can assign the function to the key.

See also: "Camera hardkey action" on page 100

Quick Action
Starts an external application, opens the graphical recall, or deletes the all measurement results, waveforms, and the history. The function is assigned to the key in "File" menu > "Frontpanel Setup" > "Hardkeys". By default, the key opens the setup dialog box.

See also: "Quick Action" on page 100

[Help]
Opens the appropriate help topic for the active tab. If no dialog box is open, the contents page of the help appears.

[Display]
Opens and closes the "Display" dialog box to configure the appearance of the waveforms, the diagram layout, color tables, and the XY-diagram. You can also assign name labels to the waveforms.

[Touch Lock]
Locks the touchscreen to prevent unintended use. When the touchscreen is off, the key is illuminated. Press again to unlock the touchscreen.

[Intensity]
Adjusts the intensity of the waveforms on the screen, or the background transparency of dialog boxes, or the transparency of result boxes. If a dialog box is open, turning the knob changes the transparency of dialog boxes. If a result box is open, the transparency of result boxes is changed. Otherwise the waveform intensity is adjusted. Press the knob to toggle between the three settings. The controlled parameter and its value are shown in the input box in the lower right corner of the screen.

3.2.3.3 Horizontal controls

The keys and rotary knobs in the Horizontal functional block adjust the acquisition settings and horizontal parameters. These settings are effective for all channel waveforms.

[Res Rec Len], [Horizontal]
Open and close the "Setup" tab in the "Horizontal" dialog box, where you can:
- Adjust the time scale, and acquisition time
- Adjust the horizontal position, and reference point
- Adjust the resolution and the record length
- Enable the roll mode

[Acquisition]
Opens and closes the "Acquisition" tab in the "Horizontal" dialog box, where you can define the acquisition processing (acquisition mode and waveform arithmetic).

[Resolution / Record Length]
The rotary knob changes the resolution or the record length. Press the knob to toggle the setting. The controlled parameter and its value are shown in the input box in the lower right corner of the screen.

For resolution, turn clockwise to increase the resolution: the time between two acquisition points gets shorter. Record length and sample rate increase while the acquisition time remains constant.

For record length, turn clockwise to increase the record length, and the resolution increases too - the time between to acquisition points gets shorter.

[Position]
The rotary knob changes the horizontal position of the waveform or the position of the reference point on the screen.

You can select if the knob changes the position or the reference point in "File" menu > "Frontpanel Setup" > "Knobs". To set the value to zero, press the knob. The current value is shown in the input box in the lower right corner of the screen.

"Horizontal position" defines the time distance of the reference point from the zero point of the diagram. Turn clockwise to move the waveform to the right.

"Reference point" defines the position of the reference point on the screen. Turn clockwise to move it to the right. The reference point marks the rescaling center of the time scale. It is indicated by a gray triangle outline at the top of the diagram. If you modify the time scale, the reference point remains fixed on the screen, and the scale is stretched or compressed to both sides of the reference point.

[Scale]
The rotary knob adjusts the time scale for all signals. The time scale is also known as timebase.

Turn clockwise to stretch the waveforms. Doing so, the scale value *time/div* decreases.

Press the knob to toggle between coarse and fine scale adjustment.

3.2.3.4 Vertical controls

The keys and knobs in the Vertical functional block select a signal and adjust the vertical scale and position of the selected signal.

[Ch <n>]
Turns on, selects, and configures a channel. If the channel is active, the key lights up in the corresponding channel color .

The effect of the keypress depends on state of the channel:
- If channel is off: Pressing the key turns on the channel and selects it.
- If the channel is on, but not selected: Pressing the key selects the channel waveform.
- If the waveform is selected: Pressing the key opens the "Vertical" dialog box for the appropriate channel.

The vertical rotary knobs are focused on the selected waveform. They are illuminated in the color of the selected waveform.

[Logic]
Opens the dialog box for configuration of parallel buses and digital channels. The key lights up if you enable at least one parallel bus. You can switch off the selected bus using the [Signal Off] key.

[Ref]
Opens the "Reference" dialog box, where you can configure and display reference waveforms. Press the key repeatedly to switch the reference waveform.

If a reference waveform is selected, the vertical rotary knobs are illuminated in white or light gray (default colors), depending on the selected waveform.

[Math]
Opens the "Math" dialog box, where you can configure the calculation of mathematical waveforms using various mathematic operations on other waveforms. Press the key repeatedly to switch the math waveform.

If a math waveform is selected, the vertical rotary knobs are illuminated in brown (default color), the brightness of the color depends on the selected waveform.

[Position] (upper knob)
The upper rotary knob changes the vertical position or the offset of the selected waveform. The horizontal axis and the selected waveform are moved vertically. The knob lights up in the color of the selected waveform.

You can select if the knob changes the position or the offset in "Menu" > "Settings" > "Frontpanel" > "Knobs" dialog.

- Position indicates the vertical location in divisions.
- Offset moves the vertical center of the selected channel to the offset value.

[Scale]
This rotary knob adjusts the vertical scale for the selected waveform. The knob lights up in the color of the selected waveform.

Turn clockwise to stretch the waveform. Doing so, the scale value V/div decreases.

Press the knob to toggle between coarse and fine scale adjustment.

[Signal Off]
Turns off the selected signal and selects the next channel, math, or reference waveform.

The key is illuminated in the color of the selected signal and changes the color according to the new selection.

3.2.3.5 Trigger controls

The keys and knob in the Trigger functional block adjust the trigger and start or stop acquisition.

[Trigger]
Opens and closes the "Trigger" dialog box, where you can:
- Select a trigger type and configure it.
- Set general trigger parameters and control the acquisition run.
- Qualify the trigger event with logic patterns.
- Configure a sequence of subsequent trigger events.
- Set up the zone trigger if option R&S RTO-K19 is installed.

[Levels]
The rotary knob sets the trigger level for all trigger types. Turn clockwise to move up the trigger level. If the selected trigger type requires two trigger levels - upper and lower level - press the knob to toggle between the two levels.

[Source]
Opens a dialog box where you can select the trigger source. Press the key again to switch the source. The key lights up in the color of the selected trigger source.

[Slope]
Toggles the trigger slope or trigger polarity, dependent on the trigger type. The current setting is shown on the trigger label, which is in the upper part of the signal bar.

[Mode]
Toggles the trigger mode between Auto and Normal. The current setting is shown on the trigger label.

[Run Stop]
Starts and stops the continuous acquisition. A green light indicates a running acquisition. A red light shows that acquisition is stopped.

[Single]
Starts a defined number of acquisitions. A green light indicates a running acquisition. A red light shows that acquisition is stopped. Press the key again to stop running acquisitions.

To set the number of acquisitions, press the [Trigger] key, select the "Ctrl/Action" tab, and set "Average count (N-single count)".

3.2.3.6 Analysis keys

The keys in the Analysis functional block provide direct access to measurement and analyzing functions. If you press [Cursor], [Zoom] or [Meas], the action starts on first keypress, and a second keypress opens the corresponding dialog box. If you press another function key, the dialog box opens.

[Cursor]
Displays vertical and horizontal cursors in the active diagram and opens the "Cursor Results" box.

Cursors are markers which are placed at points of interest on a waveform. The instrument measures the cursor positions and delta values between parallel cursors.

If you press the key while a cursor measurement is enabled, the "Cursors" dialog box opens.

In the "Cursors" dialog box, you can:
- Configure up to 4 cursor sets
- Define style and labels of the cursors

- Connect the cursor to the waveform and couple the cursors

[Meas]
Starts the default automatic measurement for the active waveform and opens the "Measurement" result box.

If you press the [Meas] key while a measurement is enabled, the "Measurements" dialog box is displayed, where you can:
- Configure amplitude and time measurements, eye, spectrum, and histogram measurements
- Configure gated measurement
- Configure long term and statistic measurements
- Configure actions to be executed if specified limits are exceeded

App Cockpit
Opens the "App Cockpit" dialog box, where you can start an application or analysis function directly, without knowing its position in the menu or toolbar.

[Zoom]
Displays a zoom diagram for the active diagram. The key is illuminated if at least one zoom is active. If you press the key while the zoom function is on, the "Zoom" dialog box opens, where you can configure several zoom areas for detailed signal observation.

[Protocol]
Opens the "Protocol" dialog box which contains the configuration of serial buses and the settings for decoding the signals.

The key lights up if the decoding of a serial bus is active. You can switch off the decoded bus using the [Signal Off] key.

[Search]
Opens and closes the "Search" dialog box, where you can:
- Configure trigger events to be searched for
- Limit the search by gating
- Configure the presentation of search results

[Masks]
Opens and closes the "Masks" dialog box. Masks are used for error detection and compliance tests of digital signals.

You can:
- Configure masks and masks segments
- Define mask test parameters
- Configure actions triggered by mask violation
- Configure the mask display

[History]
The sample memory contains several stored acquisitions before the current one, which is shown in the display. Press the key to open the quick access "History" dialog box, where you can view the stored acquisitions and use them for further analysis. Press the key again to open the main "History" dialog box with more settings and information.

The key is illuminated as long as a history acquisition or replay is displayed.

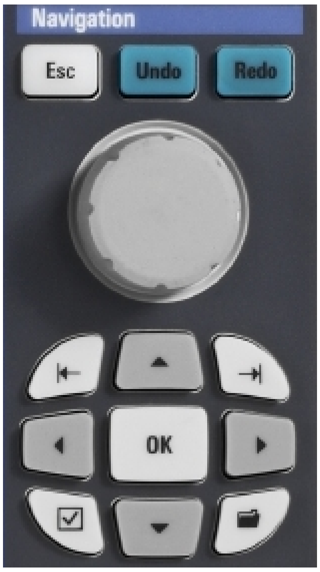
[FFT]
Opens and closes the FFT setup.

The key lights up if an FFT is active. You can switch off the FFT math waveform using the [Signal Off] key.

3.2.3.7 Navigation controls

The rotary knob and the navigation keys provide an alternative way to navigate in dialog boxes and to enter numeric data.

See also: Chapter 3.3.8, "Using dialog boxes", on page 64



[Navigation] rotary knob
The [Navigation] knob has various functions:
- In numeric entry fields: turn to increase or decrease the value.
- In tables: press to activate the edit mode, turn clockwise to increase the value or turn counterclockwise to decrease it, and press to enter the value and move to the next cell.
- To set cursor positions, histogram areas, and mask points in input boxes: press to toggle the parameter, turn clockwise to increase the value or turn counterclockwise to decrease it.
- To move zoom area, cursor line, or gate in diagrams: Turn to move the element that has the focus, and press to toggle the focus.

[Esc]
Closes a dialog box or input box.

[Undo]
Reverses the last setting actions step by step. Undo is not possible after load and recall actions, and after creating a reference waveform.

[Redo]
Recovers the undo steps in reverse order.

[OK]
The [OK] key has various functions:
- In dialog boxes and opened selection lists: the key applies the selected value.
- In tables: the key activates the edit mode. If the table cell is in edit mode, the key confirms the value, quits the edit mode and moves to the next cell.

Field left, Field right
In dialog boxes and tables, the keys move the focus.

In diagrams, they switch the focus between zoom areas, cursor lines, and gates.

Checkmark [☑]
The checkmark key [☑] has different functions depending on the focus:
- In usual dialog box: if the focus is on a selection list, the key opens the list and applies the selected value.
- In tables: activates the edit mode.

Tab
The tab key has various functions:

- In dialog boxes with only horizontal tabs, the key switches the horizontal tabs.
- In dialog boxes with horizontal and vertical tabs, the key switches the tabthat has the focus.
- In a table or diagram, the key moves the focus in the same way as the [▸] key.

Up arrow [▲], Down arrow [▼]
The up and down arrow keys have the following effects:
- In numeric edit fields: increase or decrease the parameter value.
- In tables: scroll vertically through the rows.
- In dialog boxes, for option buttons in a column: select an option. In an open selection list, the keys scroll the list.

Left arrow [◀], Right arrow [▶]
The left and right arrow keys have the following effects:
- In edit fields: move the cursor.
- In tables: scroll horizontally through the columns.
- In dialog boxes, for option buttons in a row: select an option.

3.3 Operating the instrument

There are three ways to operate the R&S RTO.

Manual operation

Use the touchscreen, keys and rotary knobs, or an optional mouse and/or keyboard. The principles of manual operation are explained in this section.

Remote control

Create programs to automatize repeating settings, tests, and measurements. The instrument is connected to a computer that runs the program.

This way of operation is described in: Chapter 23, "Remote control commands", on page 1253.

Remote operation

The remote desktop connection of Windows can be used for instrument control and file transfer. Even on computers with non-Windows operating systems, a remote desktop connection is possible using RDP applications.

For details, refer to the user manual, chapter "Remote Desktop Connection".

Remote monitoring and control of the instrument from a connected computer is also possible with a standard web browser and a LAN connection.

For details, refer to the user manual, chapter "Web Control".

Alternatively, you can use Virtual Network Computing (VNC), which requires installation of the VNC server on the R&S RTO. Installation and configuration are described in the application note "Remote Monitoring and Control of the R&S RTO with a Web Browser", available on the Rohde & Schwarz internet site.

3.3.1 Means of manual interaction

The R&S RTO provides the following means of manual interaction, which you can use alternatively or complementary:

- Touchscreen:
  Using the touchscreen is the direct interaction way. Use your finger to place waveforms on the screen, mark areas for zoom and histograms, set parameters in dialog boxes, enter data, and much more. The control elements and actions on the screen are based on common concepts, and you will easily become familiar with the user interface.

  Tapping the screen works like clicking mouse buttons:
  - Tap = click: Selects a parameter or provokes an action.
  - Double-tap = double-click has the same effect as touch and hold = right-click: Opens the on-screen keyboard or keypad, or a specific editor if available

  Use gestures to scale the waveform:
  - Spread or pinch two fingers horizontally to change the horizontal scale (timebase).
  - Spread or pinch two fingers vertically to change the vertical scale of the active waveform.
- Function keys and rotary knobs:
  The front panel provides nearly all functions and controls to operate the instrument in the classic ways, without touchscreen. As an exception, the signal bar cannot be used with front panel controls.
- Optional mouse and/or keyboard:

These devices work conform to Windows standards. The navigation keys on the front panel correspond to the keys on the keyboard.

The usage of the touchscreen and navigation keys is described in detail in the following sections.

3.3.2 Touchscreen display

3.3.2.1 Information on the display

The touchscreen of the instrument shows the waveforms and measurement results, and also information and everything that you need to control the instrument. All waveform-related display elements are shown in Figure 3-3. An overview of control elements - like dialog box, toolbar - is given in Chapter 3.3.2.2, "Control elements on the touchscreen", on page 49.

Figure 3-3: Display information

1 = Diagram
2 = Grid
3 = Trigger position
4 = Reference point (distance from trigger position to reference point = horizontal position)
5 = Trigger, horizontal and acquisition label
6 = Notifications

7 = Zoom area
8 = Zoom diagram
9 = Trigger level
10 = Signal bar with inactive waveforms (10a), minimized live waveform (10b) and signal icon with vertical settings (10c)
11 = Histogram area
12 = Histogram

Diagram

A diagram shows one or more waveforms: channel, reference, and math waveforms together with histograms, masks etc. Zoom details, XY-waveforms, spectra and other special waveforms are shown in separate diagrams.

By default, the diagram name contains the diagram number and the short names of the waveforms shown inside. To change the diagram name, touch and hold the tab name. The on-screen keyboard opens to enter the new name. Names must be unique.

To arrange the diagrams on the screen, the Rohde & Schwarz SmartGrid function helps you to find the target place simply and quickly. For details, see Chapter 3.3.5, "Rohde & Schwarz smartgrid", on page 54. You can also adjust the diagram size by dragging the diagram border.

Grid

The grid shows the vertical and horizontal divisions. The division lines are labeled with the correspondent values. The grid labels have the color of the waveform to which they belong. If several waveforms are shown in one diagram, the grid has the color of the selected waveform.

Trigger position and trigger level

The blue markers show the horizontal position of the trigger and the vertical trigger level. You can touch and move the trigger markers in the diagram to set the positions. The trigger point is the zero point of the diagram.

The trigger position can be moved outside the diagram. A red trigger position marker indicates that the trigger position is not visible.

Trigger, Horizontal, Acquisition

The "Trigger", "Horizontal" and "Acquisition" labels show the main timebase and trigger settings. If you tap a label, the relevant dialog box opens.

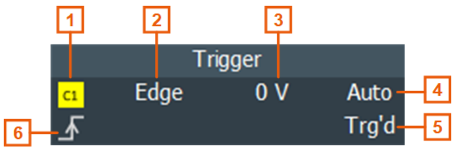


Figure 3-4: Trigger label on the toolbar

1 = Trigger source
2 = Trigger type
3 = Trigger level
4 = Trigger mode
5 = Trigger state
6 = Trigger slope

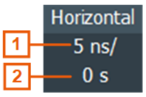


Figure 3-5: Horizontal label on the toolbar

1 = Time scale
2 = Horizontal position

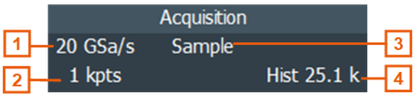


Figure 3-6: Acquisition label on the toolbar

1 = Sample rate
2 = Record length
3 = Decimation
4 = Number of acquired waveforms

Reference point
The reference point marks the rescaling center. If you modify the time scale, the reference point remains fixed on the screen, and the scale is stretched or compressed to both sides of the reference point.

You can define the position of the reference point ("Menu" > "Horizontal" > "Reference point"), and also its time distance from the trigger point of the diagram (Position knob).

Notification
The "Notification" button on the toolbar points to the status messages of the instrument. To open the message box, tap the button. See also: Chapter 3.3.10, "Instrument information and notifications", on page 68.

Zoom diagram and zoom area
Zoomed waveforms are shown in separate zoom diagrams, in addition to the waveform diagrams. On the original waveform diagram, a rectangle indicates the zoomed section of the waveform - this is the zoom area. You can modify the zoom area by dragging the rectangle as a whole, and by dragging its edges. To toggle between these modes, tap the zoom area. You can also set exact positions.

The frames of the zoom area and of the associated zoom diagram have the same color, different zooms are marked with different colors. So it is easy to assign zoom area and zoom diagram.

As for waveform diagrams, you can change the name of the zoom diagram. A zoom in a zoom and coupled zooms are also possible.

For details, see Chapter 7.1, "Zoom", on page 241.

Signal bar
The signal bar is the control center for all waveforms. All enabled waveforms are shown on the left side of the signal bar. Inactive waveforms are shown on the right side of the toolbar. Tap an inactive waveform to enable it.

Each waveform is represented by a signal icon. If the waveform is shown in a diagram, the signal icon displays its main vertical and acquisition settings. If you tap the "Minimize" icon on the signal icon, the waveform switches from the diagram area to the signal icon: the icon shows the real-time preview of the waveform. If you tap a signal icon, the dialog box with vertical settings for this waveform opens. See Chapter 3.3.4, "Working with waveforms", on page 52 for a detailed description.

In Figure 3-3, the signal icons C1 and C2 show the main settings, and the waveforms are displayed in diagrams. Other waveforms are minimized and shown in the signal icon.

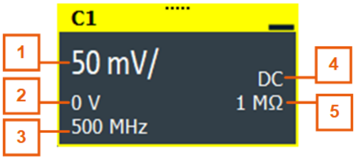


Figure 3-7: Signal label

1 = Vertical scale
2 = Offset
3 = Bandwidth
4 = Coupling
5 = Termination

If the signal bar contains many icons and not all icons are visible, touch one of the icons and scroll left or right until the required icon appears.

You can also switch off the signal bar: "Menu" > "Settings" > "Appearance" > "Diagram" > "Show signal bar".

Histogram and histogram area

A histogram shows the frequency of occurrence of voltage or time values in a bar chart directly in the diagram. The rectangular histogram area indicates the part of the waveform that is considered in the histogram. The vertical histogram counts the voltage values, and the horizontal histogram counts time values. You can switch between vertical and horizontal mode, and modify the histogram area by dragging the rectangle as a whole, by dragging its edges, or by setting exact positions.

3.3.2.2 Control elements on the touchscreen

The touchscreen provides everything you need to control the instrument, to analyze waveforms, and to get measurement results. Figure 3-8 shows the control elements on a glance.

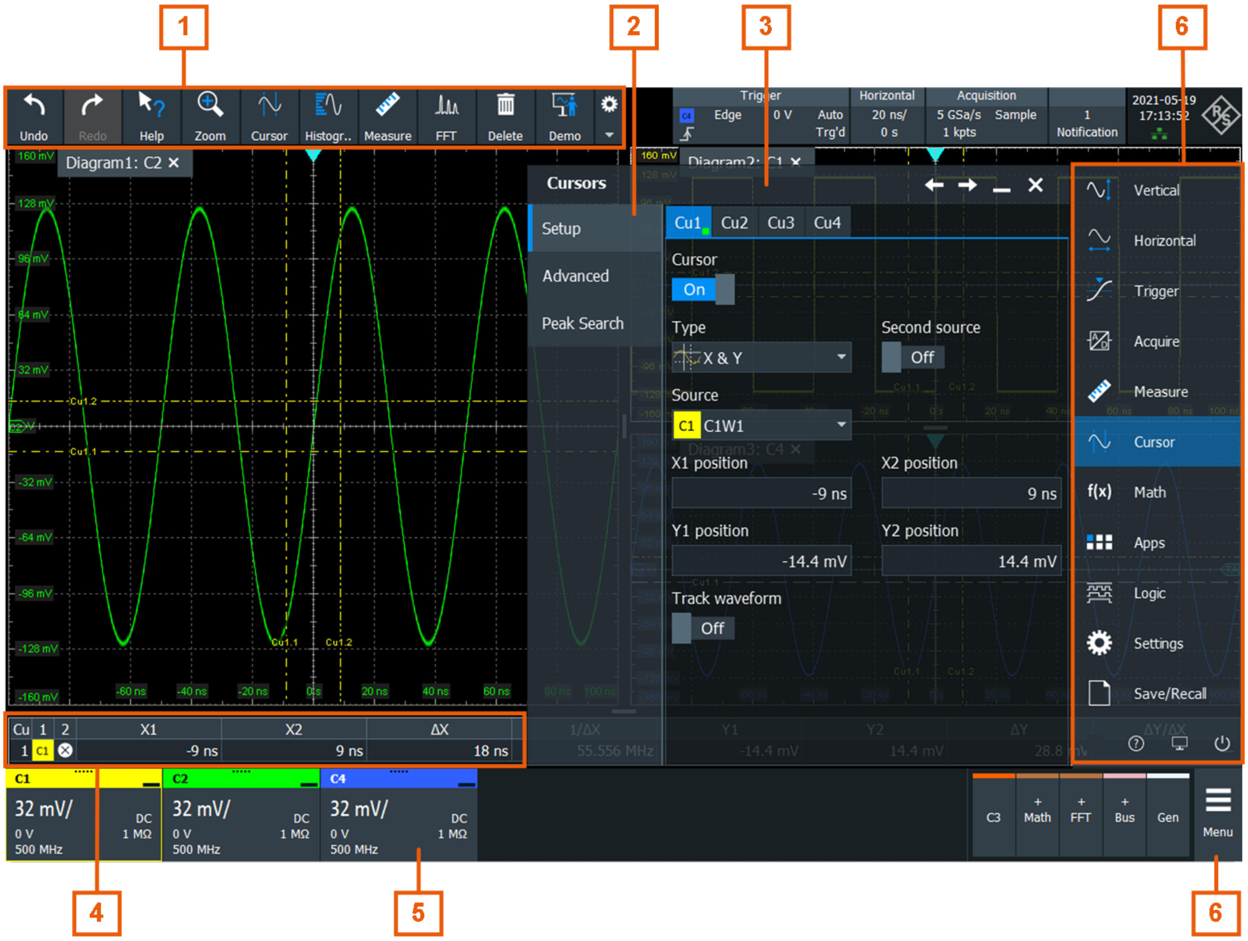


Figure 3-8: Control elements on the touchscreen

1 = Toolbar
2 = Tab in a dialog box
3 = Dialog box
4 = Result table (docked)
5 = Signal bar
6 = Menu
Not shown = Input box

Toolbar (1)

The icons on the toolbar provide quick and easy access to the most important functionality. For a detailed description, refer to Chapter 3.3.6, "Toolbar", on page 56.

If you adjust the settings of an analyzing function, e.g., cursor measurement, the toolbar assist is shown instead of the icons. The toolbar assist provides the most important settings of the current action. If you need more settings, "Advanced Setup" opens the corresponding dialog box.

Dialog box (2, 3)

The tabs of the dialog boxes contain all task-oriented settings and operations, and black buttons for calling related tabs. The usage of dialog boxes is described in Chapter 3.3.8, "Using dialog boxes", on page 64.

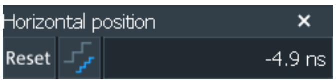
Result table (4)
If you perform cursor or automatic measurements, mask testing, or a search, the result table shows the results of the action. The position of the result table is adjustable. It can be docked (default for measurements), floating, minimized to a result icon on the signal bar, or displayed in a separate diagram on the screen.

For details, see "To define the default position of measurement results" on page 63.

Signal bar (5)
The signal bar shows all enables waveforms as described in "Signal bar" on page 48.

Menu (6)
The menu provides access to the complete functionality of the R&S RTO.

Input box
The input box appears if you adjust a value using one of the rotary knobs, or if you drag an element on the screen, for example, a cursor line. The input box shows the current value of the modified parameter. You can enter the exact numerical value, change the step size, and - if available - autoset the value directly in the input box. The box title shows the name of the currently adjusted parameter. The input box is helpful when using the multi-function rotary knobs, for example, horizontal or vertical [Position] knobs.



See also: Chapter 3.3.9, "Entering data", on page 66.

3.3.3 App cockpit

The app cockpit provides fast access to all available applications.

► To open the app cockpit:

- Open "Menu" > "Apps".
- Press the app key on the frontpanel.

3.3.4 Working with waveforms

The R&S RTO can create and display many waveform types. The most importand are:

- Channel waveforms:
  Up to three waveforms per input channel can be shown. For a four-channel instrument, 12 channel waveforms are available.
- Reference waveforms:
  Four waveforms can be used as reference for comparison and analysis.
- Math waveforms:
  Eight mathematic waveforms can be created with mathematic operations performed on channel, reference, and other math waveforms.
- Zoom waveforms:
  Show the details of waveforms.
- XY-waveforms:
  Four XY-waveforms can be created. Each XY-waveform is built from the voltage values of two source waveforms.
- Digital waveforms:
  The Mixed Signal Option R&S RTO-B1 provides 16 digital channels grouped in two logic probes (pods) with 8 channels each.

Waveform handling

The R&S RTO can show and analyze many waveforms. To handle this multitude while keeping track of it, the R&S RTO provides intelligent support:

- The color system helps to distinguish the waveforms. The color of the vertical rotary knobs indicates the signal that is focused (selected). The color of each waveform can be changed, the color of its signal icon and of the illuminated keys is adjusted to the new color. Alternatively, a color table can be assigned to a waveform.
  Settings: "Menu" > "Settings" > "Appearance" > "Colors" tab.
- Waveforms can be minimized to signal icons showing a small real-time signal view. Thus, more space in the diagram area is available without switching off waveforms.
- Diagrams are displayed on tabs – you can arrange them side by side or one above the other. To change the diagram name, double-tap the tab name.
- The Rohde & Schwarz SmartGrid function helps to arrange the diagrams.

Waveform states

Depending on its place on the screen and the effect of settings, a waveform has one of the following states:

- Off
- Active:
  The waveform is shown in a diagram
- Selected:
  One of the active waveforms that has the focus. In each diagram, one of the assigned waveforms is selected – it appears "on top" in the diagram, and the grid labels have the color of the selected waveform. Some of the toolbar functions, like cursor and histogram measurements are performed on the selected waveform. All waveform-specific settings are applied to the selected waveform of the selected diagram.
  The vertical [Position] and the [Scale] knobs are illuminated with the color of the selected waveform.
  In Figure 3-3, C1 is the selected waveform: The frame of the diagram and the signal icon are highlighted.
- Minimized:
  The waveform is shown as real-time signal view in its signal icon.

To switch a waveform on

A channel waveform is activated as soon as you connect the probe. You can switch it on and off according to your needs.

► Choose one of the following ways:
  - Press the channel key.
  - In the "Vertical" dialog box, select the channel and tap "Channel" > "On".

  The waveform is now active, selected, and is shown in the diagram.

Remote command: `CHANnel<m>:STATe` on page 1338

To select a waveform

► Choose one of the following ways:
- Tap the waveform in the waveform diagram.
- To select a channel, reference, or math waveform, press the corresponding key.
- Tap the signal icon.

Note: Zoom waveforms in zoom diagrams cannot be selected.

To minimize a waveform

► Choose on of the following ways:
- Tap the "Minimize" icon in the upper right corner of the waveform's signal label in the signal bar.
- Drag the waveform from the diagram to the signal bar.

The waveform disappears from the diagram and the minimized signal view is shown in the signal icon.

Remote command: `LAYout:SIGNal:UNASsign` on page 1306

► To set the waveform back to its previous diagram immediately, use "Undo".

To arrange a waveform using the SmartGrid

See Chapter 3.3.5, "Rohde & Schwarz smartgrid", on page 54.

To switch off a waveform

► Do one of the following:
- Select the waveform, and then press the [Signal Off] key.
- To switch off a minimized waveform, tap the "Close" icon in the upper right corner of the minimized signal view.
- Disable "Channel" in the "Vertical" > "Setup" tab.
- Tap the "Delete" icon (Recycle bin) in the toolbar, and then the waveform.
  If several waveforms overlap or lie close together, the upper (selected) waveform is switched off.

Remote command: `CHANnel<m>:STATe` on page 1338

3.3.5 Rohde & Schwarz smartgrid

The Rohde & Schwarz SmartGrid helps to create and arrange the diagrams on the screen with drag&drop. The diagram layout depends on the position where you drop the signal icon, in relation to an existing diagram.

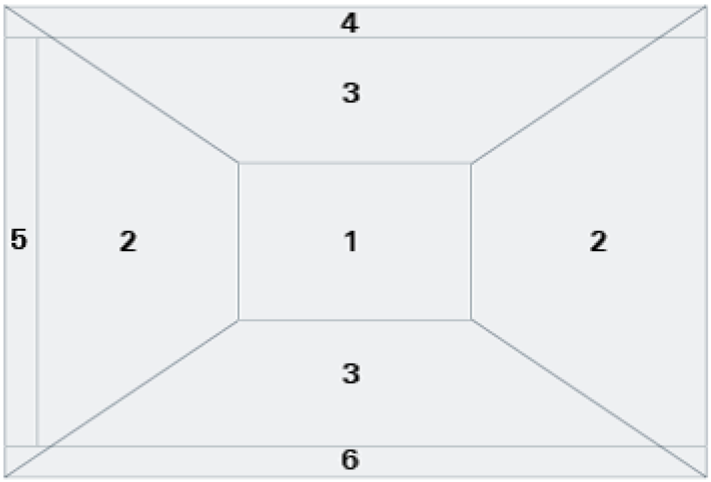


Figure 3-9: SmartGrid positions

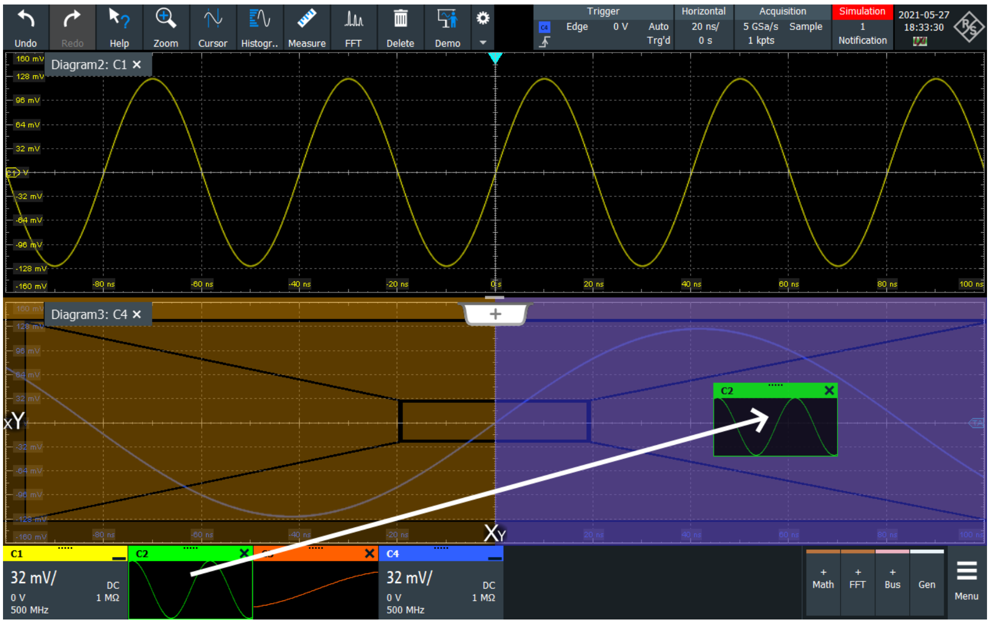
1 = In the existing diagram, overlay of signal
2 = New diagram on the left or right
3 = New diagram above or below
4 = New diagram on top of the existing diagram
5 = XY-diagram
6 = YX-diagram

The diagram configuration is deleted when you use [Preset] and `*RST`.

To arrange a waveform using the SmartGrid

You can arrange waveforms in one of the existing diagrams, or in a new diagram.

1. Drag the signal icon to the diagram area, and move it around.
   The Rohde & Schwarz SmartGrid appears and a blue area shows where the waveform will be placed.
2. Drop the waveform in the target area.
   The waveform appears in an existing or in a new diagram, and it is selected for further actions.



3. To change the size of a diagram, tap the handle between two diagram frames and drag it to the required position.

Remote commands: see Chapter 23.7.2.4, "Smartgrid", on page 1302.

3.3.6 Toolbar

The toolbar provides direct access to important control and measurement functions.

By default, the toolbar shows the most frequently used functions. You can configure the content of the toolbar, see Chapter 3.3.6.2, "Configuring the toolbar", on page 57.

3.3.6.1 Using the toolbar

Using the toolbar is easy and straightforward.

Some of the toolbar functions are one-click actions. These actions are performed immediately when you tap the icon.

Other toolbar functions are analyzing functions. These actions are interactive actions.

To use analyzing functions (interactive actions)

1. If several waveforms are shown in the diagram, select the waveform that you want to analyze.
   See: "To select a waveform" on page 54

2. Tap the icon of the function in the toolbar.
3. Check and adjust the settings in the toolbar assist.

4. To define the analyzed area, do one of the following:
   - Tap the required diagram.
   - Drag a rectangle on the diagram.

3.3.6.2 Configuring the toolbar

You can configure the content of the toolbar so that only the required functions are displayed. Furthermore, date and time can be hidden. The toolbar configuration is part of the user preferences. It is retained when you switch off and on the instrument, and you can save it in the user preferences and user-defined preset.

1. To open the toolbar configuration, tap the icon in the toolbar:

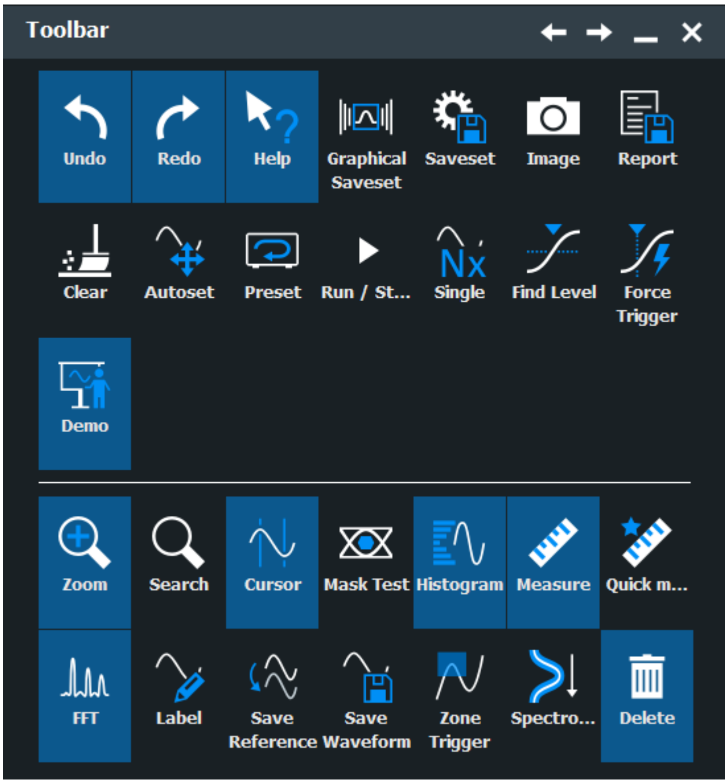
   

2. Select the required toolbar functions:
   - Disable the functions that you do not need.
   - Enable the functions that you want to add to the toolbar.

A detailed description of the toolbar functions is given in Chapter 3.3.6.3, "Toolbar functions", on page 57.

3.3.6.3 Toolbar functions

This section describes all toolbar functions in detail.



One-click actions	Interactive actions
Undo	Zoom
Redo	Search
Help	Cursor
Graphical Saveset	Mask Test
Saveset	Histogram
Image	Measure
Report	Quick meas
Clear	FFT
Autoset and Preset	Label
Run / Stop and Single	Save Reference
Find Level	Save Waveform
Force Trigger	Zone trigger
Demo	Spectrogram (option R&S RTO-K18)

You can configure the content of the toolbar, see Chapter 3.3.6.2, "Configuring the toolbar", on page 57.

The following list describes at first the default toolbar functions and then the additional functions.

Undo
Undoes the last setting actions step by step. Some actions cannot be revoked: locking the touchscreen with [Touch Lock], and saving data. The undo stack is deleted during the following actions: Reloading settings from file, and reference waveform actions (save, load and preset with active reference waveform).

Redo
Recovers the undo steps in reverse order.

Help
Enables the tooltip display. A short description appears when you tap a parameter in a dialog. To open the corresponding help topic, tap the "Show Help" button in the lower right corner of the tooltip. See also: Chapter 3.3.11, "Getting information and help", on page 69.

Graphical Saveset
Opens a window to select and load instrument settings that were previously stored in a saveset. A graphical preview helps you to find the required settings.

Saveset
Saves the current instrument settings in a saveset.

You can reload the saveset using the "Graphical Saveset" toolbar icon, or using "Menu" >"Save/Recall" > "Recall" tab > "Saveset with preview".

The filename is created according to the autonaming pattern, defined in "Menu" > "Settings" > "Save/Recall" > "Autonaming" tab.

Image
Saves a screenshot of the current display using the settings defined in "Menu" >"Save/Recall" > "Save" tab > "Screenshot".

Report
Creates a report of the current measurement settings and results using the settings defined in "Menu" > "Save/Recall" > "Save" tab > "Report".

Clear
Deletes all measurement results including long term measurement and statistics, all waveforms, and the history.

Remote command:
`DISPlay:CLR` on page 1320

Autoset and Preset
Performs an autoset, or a preset to a default state. The icons have the same functionality as the corresponding keys on the front panel. They are useful when you control the instrument remotely.


Run / Stop and Single
Starts and stops the continuous acquisition, or starts a defined number of acquisition cycles. The icons have the same functionality as the corresponding keys on the front panel. They are useful when you control the instrument remotely.

Find Level
Analyses the signal and sets the trigger level to the middle of the signal peaks.

Force Trigger
Starts an immediate single acquisition. If the acquisition is running in normal mode and no valid trigger occurs, use "Force Trigger" to confirm that a signal is available. Then you can use the displayed waveform to determine how to trigger on it.

Demo
Opens the "Demo" dialog, where you can find examples of operating the R&S RTO.

Zoom
The zoom icon on the toolbar shows the last selected zoom type. A short tap on the icon activates the selected zoom.

To use another zoom type, select it in the toolbar assist.

Standard zoom ← Zoom
Displays a magnified section of the diagram in an additional zoom diagram. It is a display zoom, instrument settings are not changed.

Touch and hold the zoom area to open the "Zoom" dialog box.

Remote command:
`LAYout:ZOOM:ADD` on page 1440

Hardware zoom ← Zoom
Changes the instrument settings - horizontal and vertical scales as well as trigger level and offset - to display a part of the diagram in greater detail.

Coupled zoom ← Zoom
Creates a coupled zoom area and its related zoom diagram. If you change the size of one zoom area, the size of all coupled zoom areas is changed as well.

Remote command:
`LAYout:ZOOM:ADDCoupled` on page 1440

Fingertip zoom ← Zoom
Magnifies the waveforms around your fingertip.

Tap the icon and put your finger on the waveform. The touched part of the waveform is displayed in a magnifier. Drag your finger on the screen to move the magnifier. You can change the zoom factor using the [Navigation] knob.

Search
Performs a search according to the settings in the "Search Setup" dialog box. Tap the icon and then tap the diagram with the waveform to be searched, or drag a rectangle to define a search gate. The search is performed on the selected waveform.

Performs a search. Tap the icon and adjust the settings in the sidebar. Tap the diagram with the waveform to be searched, or drag a rectangle to define a search gate. The search is performed on the selected waveform.

Cursor

The cursor icon shows the last selected cursor type. A short tap on the icon activates the selected cursor.

To use another cursor type, select it in the toolbar assist, and adjust the settings.

Tap the diagram where you want to set the cursors, or draw a rectangle in the diagram to position the cursor lines. The resulting cursor lines measure the selected waveform, and the results are shown. You can adjust the cursor source, type and position in the result box. Move the cursor lines by dragging them in the diagram, or by turning the navigation knob. Pressing the knob switches the parameter to be changed.

Mask Test

Starts the on-screen mask definition and the testing against the defined mask.

Tap the icon and then tap the points that build the mask. To finish the shape definition, tap "Finish segment" in the toolbar assist. To create a rectangular mask, draw a rectangle on the screen. You can add more segments to the test, and add another test. You can also move the mask segments on the screen.

To configure the mask test settings, tap the icon in the "Mask" result box.

Histogram

The histogram icon on the toolbar shows the last selected histogram type. A short tap on the icon activates the selected histogram.

To use another histogram type, select it in the toolbar assist, and adjust the source.

Tap the icon and then drag a rectangle on the diagram to mark the histogram area. The histogram for the selected waveform appears.

To change settings, touch and hold the histogram area, and the histogram toolbar assist opens.

Measure

Starts an automatic measurement.

You can run up to 8 automatic measurement groups in parallel. The "Automatic measurment" icon starts the measurement groups one after the other.

A measurement group consists of minimum one measurement, and can consist of many measurements of the same category. Tap the icon, and select the category and the measurements in the toolbar assist. Tap the diagram with the waveform to be measured. To define a measurement gate, draw a rectangle on the screen.

To modify the measurement, double-tap one of the result values.

Quick meas

Performs a set of measurements on the selected waveform or on the selected gate. You can configure up to 8 measurements to be included in quick measurement.

Tap the icon and then tap the diagram with the waveform to be measured or draw a rectangle to define the gate.

FFT
Transforms a waveform to the frequency spectrum by fast Fourier transform (FFT). The FFT trace is shown in a new diagram.

Tap the icon and adjust the settings in the toolbar assist. Tap the diagram with the waveform to be transformed. The FFT diagram is created from the selected waveform.

To adjust FFT settings, double-tap the FFT diagram.

Label
Defines a waveform label that names or explains the waveform. Tap the icon and then tap the waveform to be labeled. Enter the label text using the on-screen keyboard. The text is shown in the same color as the waveform. If you tap the display background, the label is assigned to the selected waveform. You can drag the label to another position.

Save Reference
Copies the selected source waveform with all its settings to the reference waveform. If the acquisition is running, the reference waveform is a snapshot. You can configure up to four reference waveforms.

Tap the icon, and select the required reference waveform (R1 to R4) in the toolbar assist. Tap the waveform to be copied as reference.

Save Waveform
Exports the waveform data to file using the settings defined in "Menu" > "Save/Recall" > "Save" tab > "Waveform" > "Setup" tab.

The filename is created according to the autonaming pattern.

Tap the icon and then tap the waveform to be exported. If you tap the display background, the selected waveform is exported, or a multichannel export is performed if configured.

Zone trigger
Defines a zone trigger, which combines the trigger condition with the intersection or non-intersection of one or more zones or masks.

Tap the icon and then tap the corner points of the zone on the screen. Tap "Finish zone". You can add more zones to the trigger condition.

Spectrogram (option R&S RTO-K18)
Starts an FFT and the spectrogram. The FFT trace and the spectrogram are shown in separate diagrams.

Tap the icon. Select the source in the sidebar, or tap diagram with the waveform to be transformed. The diagrams are created from the selected waveform.

Delete
Removes waveforms, measurements, zoom, histograms, mask segments and other elements from the display.

Tap the icon and then tap the element to be deleted, or the waveform to be switched off.

3.3.7 Displaying results

The results of measurements, mask tests, searches, protocol decoding and others are displayed immediately. There are several places to display the results:

- In a table that is docked below the diagram (not available for mask test results)
- In a floating result box in front of the diagrams, which you can move on the display
- In a minimized view (result icon) on the signal bar
- In a separate that can be arranged similar to a diagram

The default position and the font size can be adjusted.

To define the default position of measurement results

For results of automatic, cursor measurements and protocol measurements, the docked position below the diagram is the initial default position.

To change the default position of the results:

► Open the "Menu" > "Settings" > "Appearance" dialog.

- Automatic measurements: select "Measurement" tab > "Result position".
- Cursor measurements: select "Cursor" tab > "Result position".
- Protocols: select "Protocol" tab > "Result position".

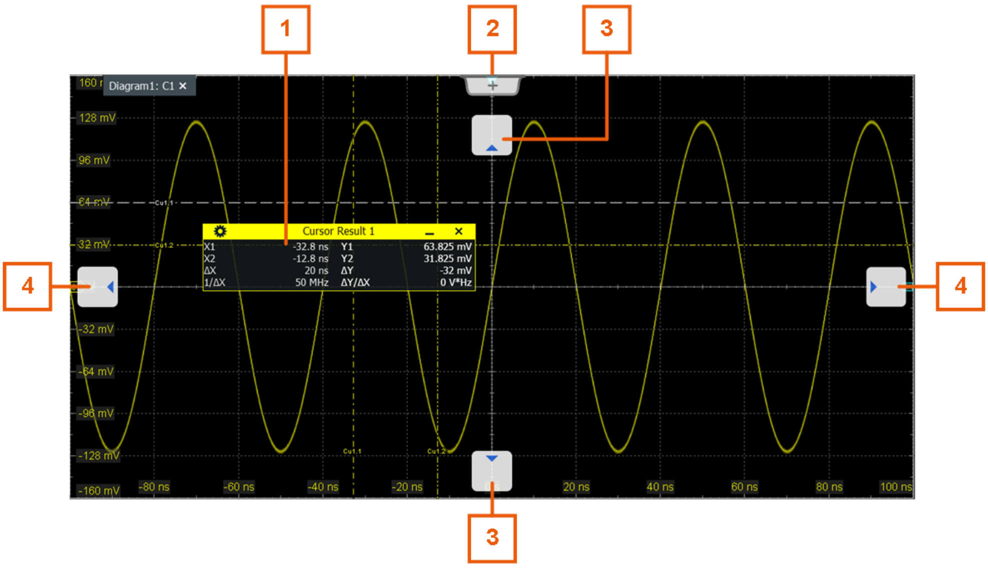
To arrange the results on the display

1. If the results are listed in a docked table, touch and hold a result line and drag it.

   The results are now shown in a result box.

   Note: Once you undock the measurement results, you cannot move the result box back to the docked position.

2. Touch and hold the title of the result box and drag the box on the screen. The SmartGrid indicates where the result box will be placed.
   - If you drop the box on one of the buttons, the results are shown in a separate tab besides, above, or below the diagram.
   - If you drop the box on the signal bar, a result icon is created.
   - If you drop the box somewhere else, a floating result box is placed.



1 = Floating result box
2 = New tab
3 = Table in a tab above or below
4 = Table in a tab on the left or right

To open the corresponding setup dialog box

1. Double-tap one of the result values.
   If a result box is shown, you can also tap the ⚙ icon.
2. In the toolbar assist, tap "Advanced Setup" .

   The dialog box with corresponding settings opens.

To adjust the font size of results

1. Open the "Menu" > "Settings" > "Appearance" dialog.
2. Select the "Dialogs" tab.
3. Set the "Result dialog" > "Font size".

3.3.8 Using dialog boxes

All functionality is provided in dialog boxes as known from computer programs. You can control the instrument intuitively with the touchscreen. This section provides an overview of the accessing methods and describes how to use the dialog boxes.

Each dialog box has four icons in the upper right corner:

	Go back: opens the previously opened dialog box.
	Go forward: opens the next dialog box.
	Minimizes the dialog box to a small box that only contains the last selected function.
	Closes the dialog box.

For direct access to important control and measurement functions, use the toolbar, see Chapter 3.3.6, "Toolbar", on page 56.

To open a dialog box

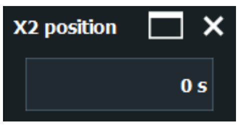
► Perform one of the following actions:
- Open the "Menu", and select the menu entry.
- Press the function key on the front panel.
- Double-tap a result icon, or tap the icon in a result box to open the corresponding dialog box.
- To open the "Vertical" dialog box of a waveform, tap the signal icon.
- Tap the "Horizontal", "Acquistion" or "Trigger" label to open the corresponding dialog box.

To minimize a dialog box

If you want to change only one setting during analysis, and you need to change it often, you can display a small box that only contains the required setting.

1. Tap the setting that you need to modify.
2. Tap the "Minimize" icon in the upper right corner of the dialog box.

The dialog box turns into a small box that contains only the setting, that was last selected.



3. To restore the complete dialog box, tap the "Maximize" icon in the small box.

To close a dialog box

► Tap the "Close" icon in the upper right corner.

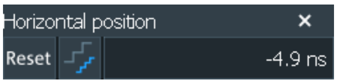
3.3.9 Entering data

To set parameter values and enter other data, you use the various knobs and the on-screen keypad or keyboard.

Using scale, position and level knobs

The instrument has dedicated rotary knobs to set vertical and horizontal positions and scale, and the trigger level. When you turn a knob, the input box appears in the lower right corner of the screen, showing the parameter name and current value.

1. Turn the knob to change the value.
2. Press the knob:
   - [Scale]: to toggle the increment.
   - [Position]: to set to zero.
   - [Levels]: to set the trigger level to 50% of the signal.
3. The input box also provides settings
   - Tap the "Steps" icon to toggle the increment.
   - Tap the "Reset" icon to set the parameter to the autoset value (if available).



The [Navigation] knob is described in Chapter 3.2.3, "Keys and controls", on page 36.

To enter values with the on-screen keypad

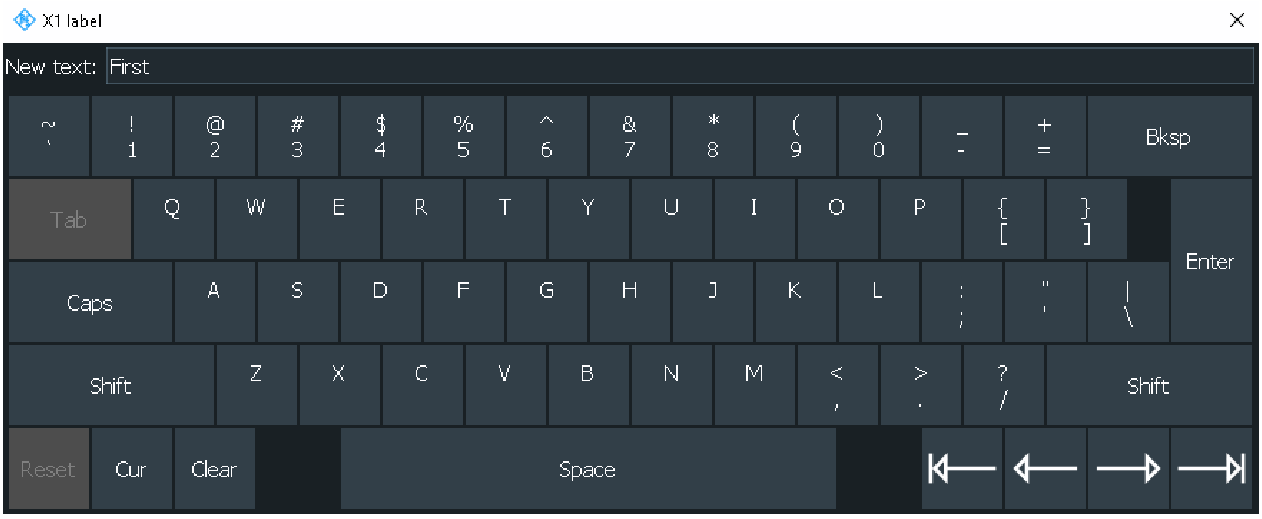
For data input in dialog boxes, the touchscreen provides an on-screen keypad to enter numeric values and units. For text input, the on-screen keyboard with English key layout is used.

1. Double-tap the entry field.
   The on-screen keypad opens.

2. Enter the numeric value using the following methods:
   - To use the default value, tap "Reset" (if available).
   - To use the minimum or maximum value, tap "Min" or "Max", respectively.
   - To increase the displayed value in fixed steps, tap "Inc".
     To decrease the value in fixed steps, tap "Dec".
     To toggle between small steps and large steps, tap the "Steps" icon.
   - To get the value that was used before the keypad was displayed, tap "Cur".
   - To enter a user-defined value, tap the numbers and complete the entry by tapping the unit button.
     – The arrow buttons move the cursor left or right.
     – "Bksp" deletes the last character before the cursor.
     – "±" changes the sign of the value.

To enter data with the on-screen keyboard

1. Double-tap the entry field to open the on-screen keyboard.
   If available, you can also tap the keyboard icon on the right of the entry field.



2. Enter the text as you would on a normal keyboard.
   - To enter a series of capital characters, tap "Caps".
     To enter one capital character, tap "Shift".
   - To use the currently defined value, tap "Cur". This is the value that was used before the keyboard was displayed.
   - The arrow buttons move the cursor left or right.
   - "Bksp" deletes the last character before the cursor.
3. Tap "Enter" to complete the entry.

To enter numeric data in a dialog box with navigation controls

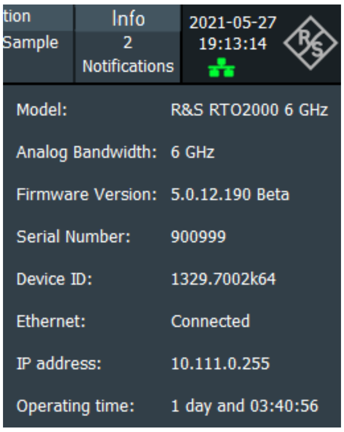
1. To navigate to the entry field, press the [⇤] and [⇥] keys.
2. To change the value with a small step size, turn the rotary knob.
   Alternatively, press the [▲] and [▼] keys for a larger step size.

If you edit numeric data in tables, the entry field must be in edit mode. To activate the edit mode, press ENTER, or the [☑] key, or the navigation rotary knob.

3.3.10 Instrument information and notifications

In the upper right corner of the screen, you see the Rohde & Schwarz logo, date and time, the symbolic information on LAN connection and the notifications status.

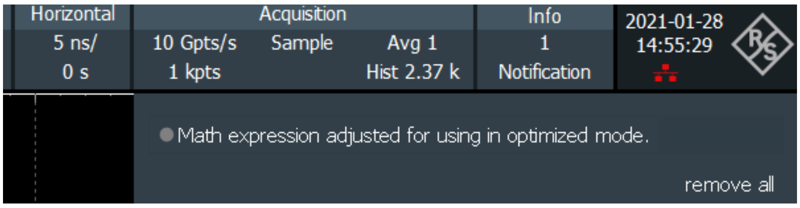
► To see the instrument information, tap the Rohde & Schwarz logo.



Notifications are status messages, information on mismatching settings and similar information. They are displayed for a few seconds and saved.

The color of the dot before the text indicates the severity: gray for information, orange for warnings, and red for errors.

► To read the notifications, tap "Notification".
You can also delete the list of notifications.



1. To hide the date and time, open "Menu" > "Settings" > "Appearance" > "Diagram" tab.
2. Disable "Show date / time".

3.3.11 Getting information and help

If you need information on the instrument's functionality, you can use the following sources:

- Tooltips give a short description of the parameter.
- The context help provides functional description on a setting, and the corresponding remote command.
- The general help opens the help window with a table of contents, where you can browse and search for information.
- Videos explain various analyzing and measurement tasks.
They are available in the R&S®RTO2000 - Media Center.

3.3.11.1 Displaying help

To display tooltips and context help

1. Enable the "Help" icon on the toolbar.

2. Tap the parameter for which you need information.

   The tooltip opens.

3. To open the corresponding help topic, tap the "Show Help" button in the lower right corner of the tooltip.

   The "Help" window opens and displays the comprehensive description and the corresponding remote command. You can browse the help for further information.

   Note: The tooltip icon disables automatically when you tap a parameter. To show another tooltip, tap the tooltip icon again.

To open general help

1. Open the "Menu".
2. Tap the "Help" icon.

   The help window opens with the "Contents" page, where you can select the topics.

3.3.11.2 Using the help window

The Help window contains several tabs:

- "View" - shows the selected help topic
- "Contents" - contains a table of help contents
- "Index" - contains index entries to search for help topics
- "Search" - provides text search

The Help toolbar provides some buttons:

- To browse the topics in the order of the table of contents: Up arrow = previous topic, Down arrow = next topic
- To browse the topics visited before: Left arrow = back, Right arrow = forward
- To increase or decrease the font

To navigate the Help, use the touchscreen.

To search for a topic in the index

The index is sorted alphabetically. You can browse the list, or search for entries in the list.

1. Switch to the "Index" tab.
2. Select the "Keyboard" icon besides the entry field.
3. Enter the first characters of the keyword you are interested in.
   The entries containing these characters are displayed.
4. Double-tap the suitable index entry.
   The "View" tab with the corresponding help topic is displayed.

To search topics for a text string

1. Switch to the "Search" tab.
2. Select the "Keyboard" icon besides the entry field.
3. Enter the string you want to find.
   If you enter several strings with blanks between, topics containing all words are found (same as AND operator).
   For advanced search, consider the following:
   - To find a defined string of several words, enclose it in quotation marks. For example, a search for *"trigger qualification"* finds all topics with exactly *"trigger qualification"*. A search for *trigger qualification* finds all topics that contain the words *trigger* and *qualification*.
   - Use "Match whole word" and "Match case" to refine the search.
   - Use operators AND, OR, and NOT.

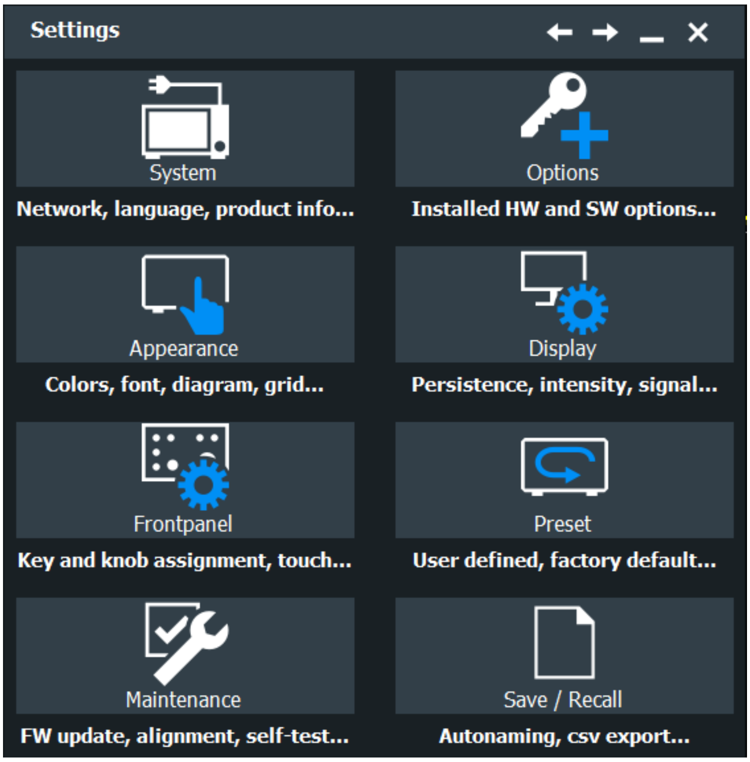
To close the Help window

► Select the "Close" icon in the upper right corner of the help window.
Or: Press the [Esc] key.

4 Instrument setup

Access: "Settings" menu > "Settings" dialog.

In the "Settings" dialog, you can adapt various instrument settings to your requirements, such as language, display appearance, and assign functions to some keys.



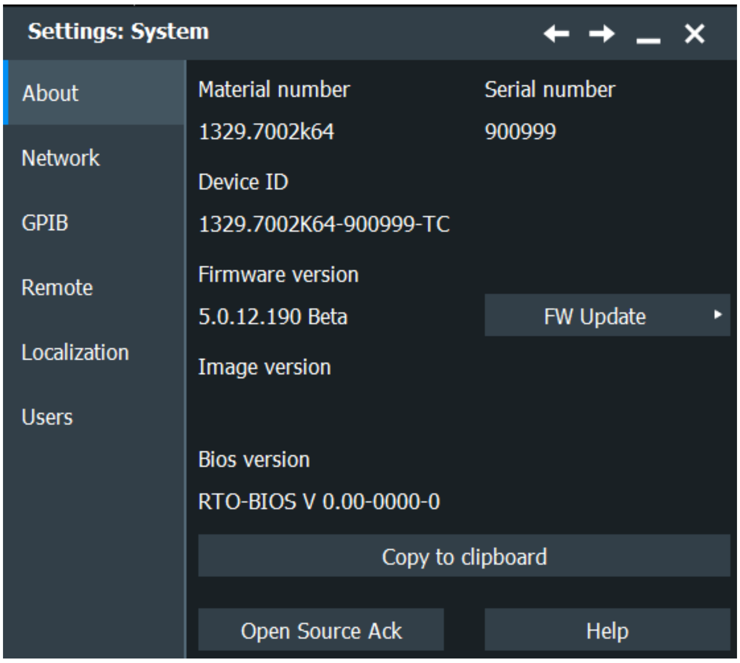
The following settings and procedures are described in the current chapter:

4.1 System settings

In the "Settings" > "System" dialog box, you find all instrument, firmware and network related information. Here you can also set the language that is used in the dialogs.

4.1.1 About settings

Access: "Menu" > "Settings" > "System" > "About"



FW Update
Opens the Firmware update dialog box.

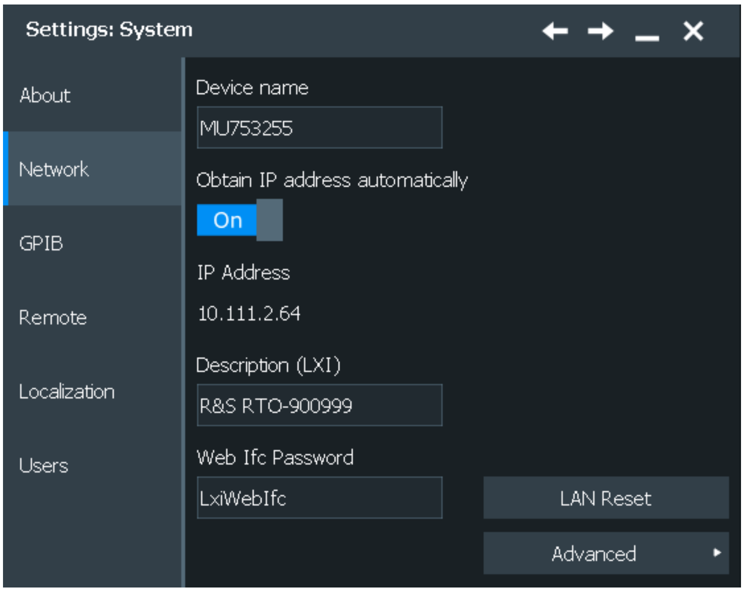
Copy to clipboard
Copies the instrument information, which you see in the dialog box, to the clipboard.

Open Source Ack
Opens the open source acknowledgment document, which provides verbatim license texts of the used open source software.

Help
Opens the help window with a table of contents, where you can browse and search for information.

4.1.2 Network settings

Access: "Menu" > "Settings" > "System" > "Network"



Device name
Indicates the currently defined computer name, the defined IP address and DHCP address enabling. These values are required to configure the instrument for work in a network.

NOTICE! Risk of network problems. All parameters can be edited here; however, beware that changing the device name has major effects in a network. For details, see Chapter 22.2, "Setting up a network (LAN) connection", on page 1226.

Remote command:
`DIAGnostic:SERVice:COMPutername` on page 1298

Obtain IP address automatically
If enabled, the IP address of the oscilloscope is obatained automatically.

IP Address
Indicates the currently defined computer name, the defined IP address and DHCP address enabling. These values are required to configure the instrument for work in a network.

Description (LXI)
Instrument description of the R&S RTO.

Web Ifc Password
Password for LAN configuration. The default password is *LxiWebIfc*.

LAN Reset
Resets the LAN configuration to its default settings using the network configuration reset mechanism (LCI) for the instrument. The following parameters are reset:

Parameter	Value
TCP/IP mode	DHCP + auto IP address
Dynamic DNS	Enabled
ICMP ping	Enabled
Password for LAN configuration	LxiWebIfc

The LAN settings are configured using the instrument's web browser.

Advanced
Opens the standard Windows "Network Connections" dialog box to configure a network connection. Only users with administrator rights can fulfill this task.

4.1.3 GPIB settings

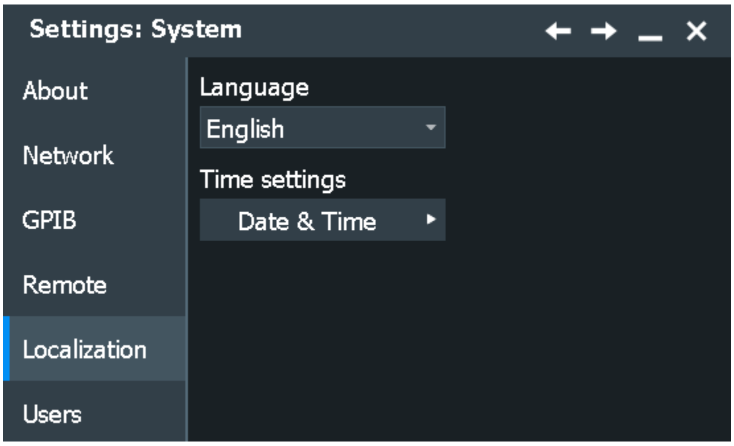
For a description of the GPIB settings, see: Chapter 22.5.2.2, "Remote settings", on page 1241.

4.1.4 Remote settings

For a description of the remote settings, see: Chapter 22.5.2.2, "Remote settings", on page 1241.

4.1.5 Localization settings

Access: "Menu" > "Settings" > "System" > "Localization"



Language
Selects the language in which the dialog boxes, result boxes and other screen information is displayed.

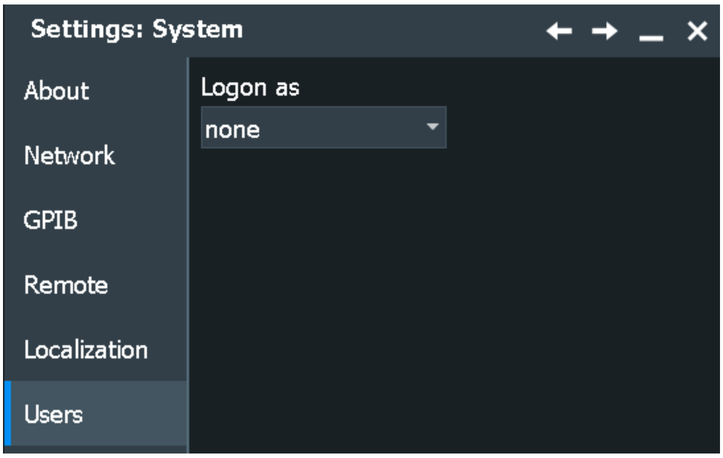
Time settings
Opens the standard Windows "Date and Time Properties" dialog box to set the correct date and time. Only users with administrator rights can fulfill this task.

Note: Usually date and time are set correctly. To adjust your regional time, select the correct time zone rather than changing the time.

Remote command:
`SYSTem:DATE` on page 1297
`SYSTem:TIME?` on page 1297

4.1.6 Users settings

Access: "Menu" > "Settings" > "System" > "Users"



Log on as
Sets the user that is automatically logged on during the startup process of the instrument. The change of this setting takes effect at the next instrument startup.

See Chapter 22.1.1, "Logon", on page 1222 for restrictions of the standard user and how to change the auto-logon.

"User autologon"	Auto-logon as standard user with limited access. Enter the "User name": *NormalUser* and the "Password" of the standard user.
"Admin autologon"	Auto-logon with unrestricted access to the instrument and network. The setting is only available for administrators. Enter the "User name": *Instrumen* and the administrator's "Password" to enable the auto-logon.
"None"	No auto-logon, user name and password are requested at instrument startup.

4.2 Option settings

Additional options for the R&S RTO can be enabled using a license key. To obtain the license key, consult your sales representative.

The license type defines the duration of applicability and the portability of a license. The following license types are provided: evaluation, permanent, portable, quantified, timed with duration of 1, 3, 6 or 12 months. A license can also be in the states deactivated and expired.

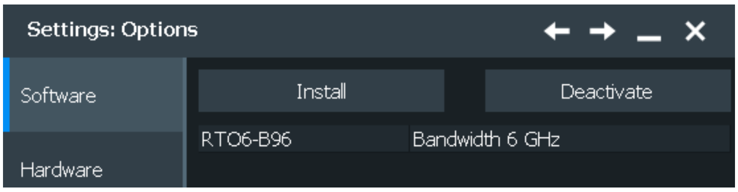
Unregistered licenses

Unregistered licenses are not assigned to a particular instrument. The instrument accepts only registered licenses. If your license is delivered unregistered, use the online tool R&S License Manager to register the license for your instrument. The registration of a permanent license is irreversible, so ensure that you register it for the correct instrument. The address of the tool is https://extranet.rohde-schwarz.com/service.

The R&S License Manager also allows you to move a portable license to another instrument.

4.2.1 Software options settings

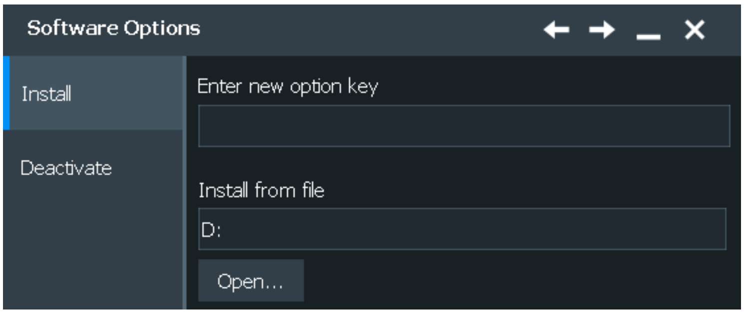
Access: "Menu" > "Settings" > "Options" > "Software"



In this dialog, you can access settings for installing and deactivating options.

4.2.1.1 Install options

Access: "Menu" > "Settings" > "Options" > "Software" > "Install"



In the "Install" tab, you can install new options or deactivate existing options using license keys.

Enter new option key

Enter the license key here to activate the option. For license keys delivered as a file, use Install from file. Only users with administrator rights can activate options.

Install from file
If you got a license file, install the license here.

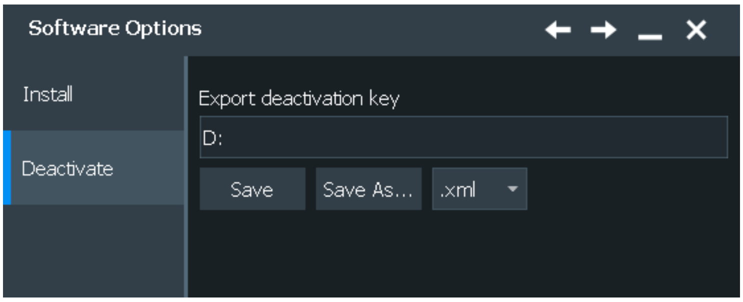
Tap "Open" to open the file selection dialog, or enter the complete path and filename. For details, see Chapter 12.5, "File selection dialog", on page 478. Only users with administrator rights can activate options.

When you move a portable license, use this function to import the deactivation key that is generated by the "R&S License Manager". See also Chapter 4.2.1.3, "Using a license server", on page 78.

4.2.1.2 Deactivate options

Access: "Menu" > "Settings" > "Options" > "Software" > "Deactivate"

In the "Deactivate" tab, you can export the deactivation response.



Export deactivation key
When you move a portable license, or deactivate an option, you have to note the response key, or to save the response to a file. The "R&S License Manager" needs the response key.

4.2.1.3 Using a license server

Software licenses can also be provided by the R&S®License Server. In this case, all available licenses are stored on a specified server. When you need a particular application, you can obtain a license from the server. When you no longer need the license, you return it to the server and it becomes available to other users again. These licenses are also referred to as *floating licenses*, as opposed to permanent licenses.

The R&S®License Server must be set up as described in the R&S®License Server - Managing Floating Licenses - User Manual.

You can find the manual in the Windows menu of your instrument: "Start > All Programs > R&S License Server > Open License Server Manual".

To open the License Server

► In the Windows notification area, select , and then "Open Manager".

a) Optional: If the status icon (, or) in the Windows notification area is not shown yet, click "Start > All Programs > R&S License Server > License Server Manager"".
   The icon is now available in the Windows notification area.
b) Optional: If the status is (stopped), select the icon, and then select "Start License Server". Then, select > again.

The license server opens in the default web browser. The program adds all available Rohde & Schwarz products (smart cards or devices) to a list and automatically shows the first product that is detected. By default, the "Licenses" list opens with an initial filtering for showing only active licenses.

4.2.2 HW options settings

This tab informs about the availability of hardware options.

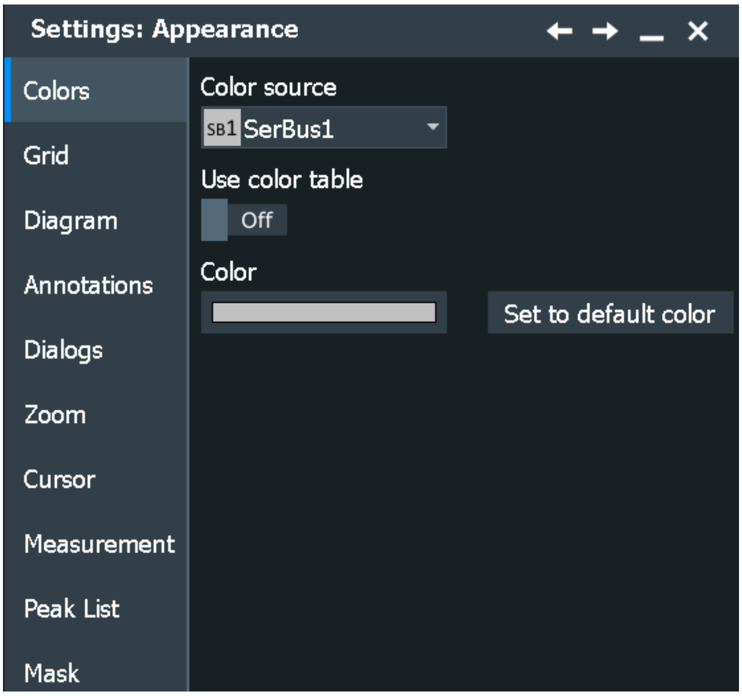
4.3 Appearance settings

In the "Settings" > "Appearance" dialog box, you define the look and feel of the display element, e.g. waveform colors, result position, or grid behavior.

4.3.1 Colors settings

Access: "Settings" > "Appearance" > "Colors"

In this dialog, you can define the colors to be used for the different waveforms.



Color source

Selects the waveform to which the color table and the labels are assigned.

A spectrogram (option R&S RTO-K18) always has the same color as the math (spectrum) waveform from which it is created.

Use color table

If enabled, the selected waveform is displayed according to its assigned color table.

If this option is disabled, the selected color is displayed, and the intensity of the specific signal color varies according to the cumulative occurrence of the values.

Remote command:
`DISPlay:COLor:SIGNal<m>:USE` on page 1299

Color

Shows the current color of the selected waveform. To change the color, tap the button and select a color. The color of the waveform, its signal icon, channel icon, and of the illuminated keys is adjusted to the new color.

Remote command:
`DISPlay:COLor:SIGNal<m>:COLor` on page 1298

Assigned color table

Adjust the waveform colors to suit your preferences. For each of the following waveform types you can assign a suitable color table:

- Analog and digital channels
- Reference waveforms
- Results of a mathematical function, also for FFT and derived spectrogram.
- Measurements and tracks
- XY-traces

- Serial buses if a protocol option is activated
- Parallel buses if MSO option is installed

See also: Chapter 4.3.2, "Color tables", on page 81

Remote command:
`DISPlay:COLor:SIGNal<m>:ASSign` on page 1298

Set to default color
Resets the color of the selected waveform to the factory default.

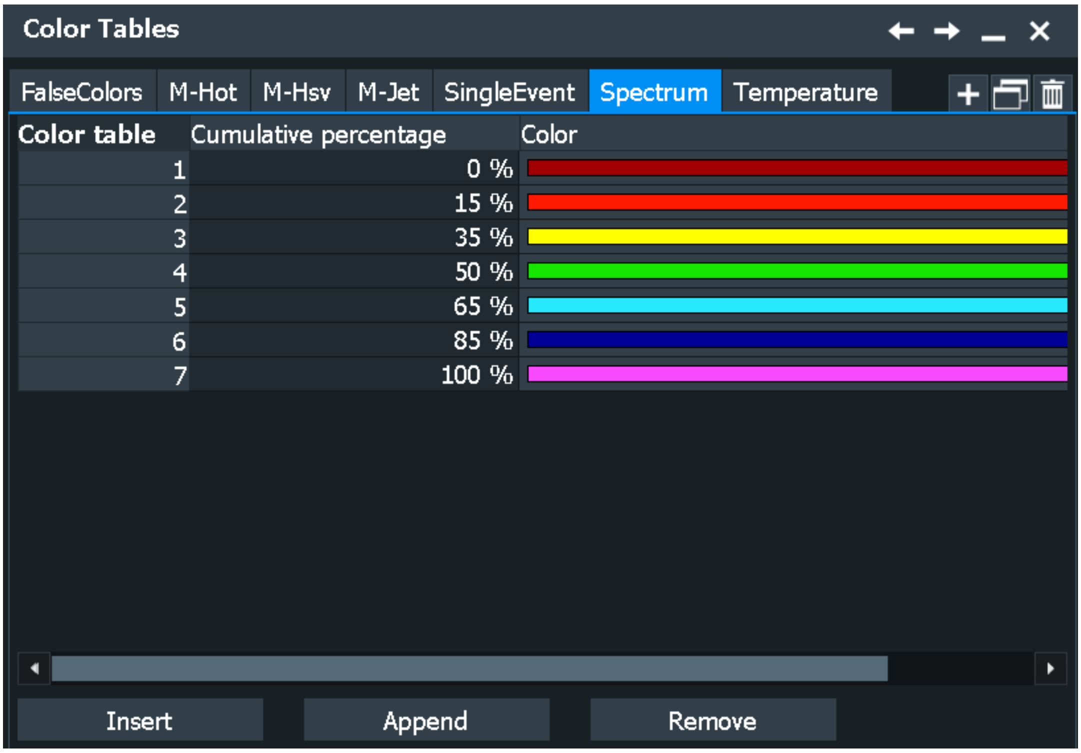
4.3.2 Color tables

Access: "Settings" > "Appearance" > "Color" > "Edit Color Table"

Color tables define the color of the waveform pixels depending on the cumulative occurrence of the associated values. By default, the intensity of the specific waveform color varies according to the cumulative occurrence of the values. The more often a value occurs, the darker the color of the data point is displayed.

The following default color tables are provided:

- "False colors"
- "M-Hot"
- "M-Hsv"
- "M-Jet"
- "Spectrum"
- "Single Event"
- "Temperature"



The editing table allows you to edit existing color tables or add new ones that can then be assigned to the waveforms.

See also:
- "Assigned color table" on page 80

Remote commands

The following remote commands are used to configure color tables:

`DISPlay:COLor:PALette:COUNt?` on page 1299

`DISPlay:COLor:PALette:ADD` on page 1299

`DISPlay:COLor:PALette:REMove` on page 1299

`DISPlay:COLor:PALette:POINt:INSert` on page 1300

`DISPlay:COLor:PALette:POINt:ADD` on page 1300

`DISPlay:COLor:PALette:POINt[:VALue]` on page 1300

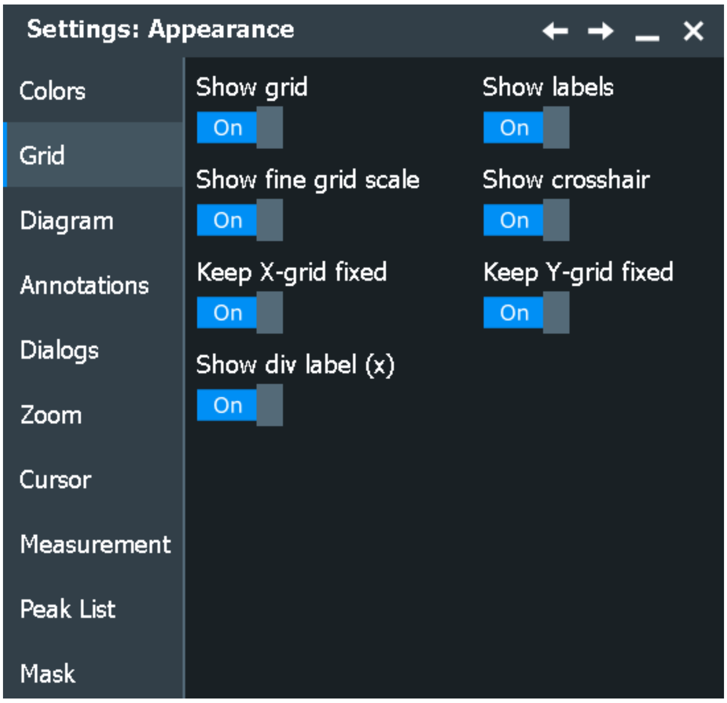
`DISPlay:COLor:PALette:POINt:COUNt?` on page 1301

`DISPlay:COLor:PALette:POINt:REMove` on page 1300

`DISPlay:COLor:PALette:COUNt?` on page 1299

4.3.3 Grid appearance settings

Access: "Settings" > "Appearance" > "Grid"



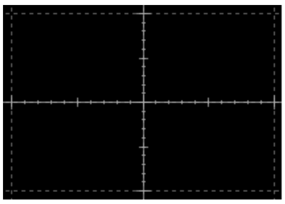
Show grid
If selected, a grid is displayed in the diagram area. A grid helps you associate a specific data point to its exact value on the x- or y-axis.

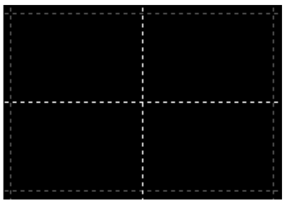
Remote command:
`DISPlay:DIAGram:GRID` on page 1301

Show labels
If selected, labels mark values on the x- and y-axes in specified intervals in the diagram.

Remote command:
`DISPlay:DIAGram:LABels` on page 1301

Show fine grid scale
If selected, the crosshair is displayed as a ruler with scale markers. If disabled, the crosshair is shown as dashed lines.





Remote command:
`DISPlay:DIAGram:FINegrid` on page 1301

Show crosshair
If selected, a crosshair is displayed in the diagram area. A crosshair allows you to select a specific data point by its co-ordinates.

Remote command:
`DISPlay:DIAGram:CROSshair` on page 1302

Keep X-grid fixed
If enabled, the vertical grid lines remain in their position when the horizontal position is changed. Only the values at the grid lines are adapted.

Remote command:
`DISPlay:DIAGram:XFIXed` on page 1302

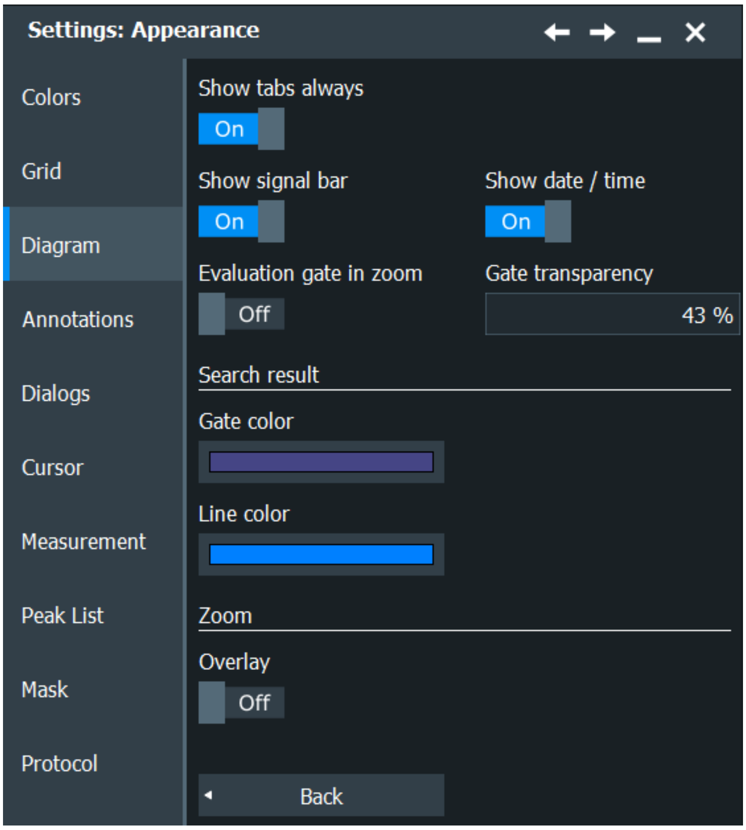
Keep Y-grid fixed
If enabled, the horizontal grid lines remain in their position when the position of the curve is changed. Only the values at the grid lines are adapted. Fixed horizontal grid lines correspond to the behavior of traditional oscilloscopes.

Remote command:
`DISPlay:DIAGram:YFIXed` on page 1302

Show div label (x)
If selected, the time scale value is shown at the diagram bottom instead of the horizontal grid labels. For example, 10 ns/div is shown instead of the values 0, 10, 20, 30... ns.

4.3.4 Diagram appearance settings

Access: "Settings" > "Appearance" > "Diagram" tab



Show tabs always
If selected, the tab titles of all diagrams are displayed: "Diagram1", "Diagram2" ...

If cleared, the tab titles are not shown except for titles in a tabbed diagram. In tabbed diagrams, the tab titles are required to change the tabs.

Remote command:
`DISPlay:DIAGram:TITLe` on page 1306

Show signal bar
If enabled, the signal bar is displayed on the right of the diagram area.

The signal bar contains signal icons that display either real-time views of minimized waveforms, or the main settings of displayed waveforms. On the top of the bar, the timebase label and trigger label provide general information for all displayed channels.

Remote command:
`DISPlay:SIGBar[:STATe]` on page 1306

Show date / time
If enabled, the date and time are shown near to the Rohde & Schwarz logo at the upper right corner of the screen.

Evaluation gate in zoom
If enabled, the available histogram areas, masks, and measurement gates are shown in the zoom diagrams. If the evaluation gate is within the zoom area, the display helps to move or modify the evaluation gates in the zoom window.

Make sure that the option is disabled if the zoom area and the evaluation gate are of nearly the same size to avoid conflicts in operation.

Gate transparency

Sets the transparency of the area that is defined as measurement or search gate. The setting only takes effect if "Show gate" is enabled.

Remote command:
`DISPlay:GATE:TRANsparency` on page 1306

Search result gate symbol color

Sets the color of the search zoom area. The search zoom area is displayed if "Show search zoom windows" is enabled.

See also: Chapter 11.4.2, "Display zoom settings", on page 437.

Search result line color

Sets the color of the search result markers. The markers are displayed if "Show search zoom windows" is enabled.

Zoom overlay

Shows all zooms of a diagram in one zoom window. The zoomed areas are overlaid for better comparison of the zoomed waveforms.

The setting affects all zoom diagrams.

See also: Chapter 7.1, "Zoom", on page 241

Remote command:
`LAYout:ZOOM:ONEDiagram` on page 1307

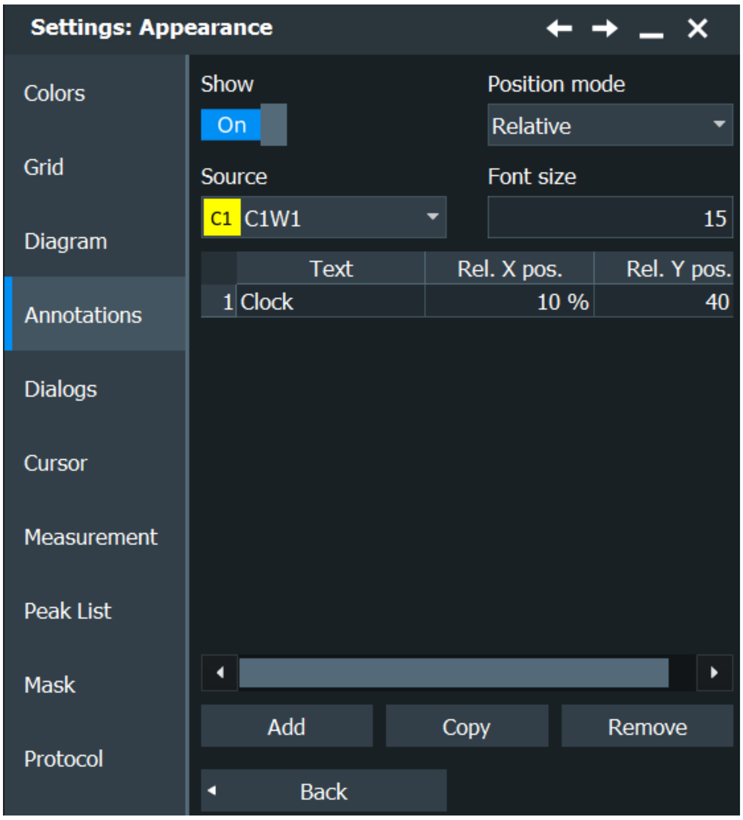
4.3.5 Annotations appearance settings

Access: "Settings" > "Appearance" > "Annotations" tab

Using labels, you can annotate the waveforms to name or explain each waveform. The text is shown in the same color as the assigned waveform. Each label has its individual position.

You can enter exact positions in the dialog box, or drag the labels on the screen to the required position. The position can be a fixed one (relative to the screen), or a flexible position (absolute, assigned to the axes).

To add labels quickly, you can add the "Label" icon to the toolbar and use it.



Show / Source

Selects the souce of the label and enables the display.

Remote command:

`DISPlay:SIGNal:LABel:SHOW` on page 1310

Position mode

Defines the label position either relative to the diagram or with absolute values according to the units of the waveform. Relative positions are fixed, whereas absolute positions move with the waveform display when the scales, the vertical position or offset, or the reference point are changed.

The position mode applies to all labels of the selected waveform. For different waveforms, different position modes can be selected.

"Relative"	Sets a fixed position in percent of the screen counting from the upper left corner.
"Absolute"	Sets the position in time and voltage values, or in other units depending on the waveform character.

Remote command:

`DISPlay:SIGNal:LABel:POSMode` on page 1307

Font size

Defines the size of the labels in the diagram.

Remote command:

`DISPlay:SIGNal:LABel:FONTsize` on page 1308

Labels

For each waveform, the "Labels" table shows the assigned texts and their positions. Enter the label text and the horizontal and vertical positions for each label.

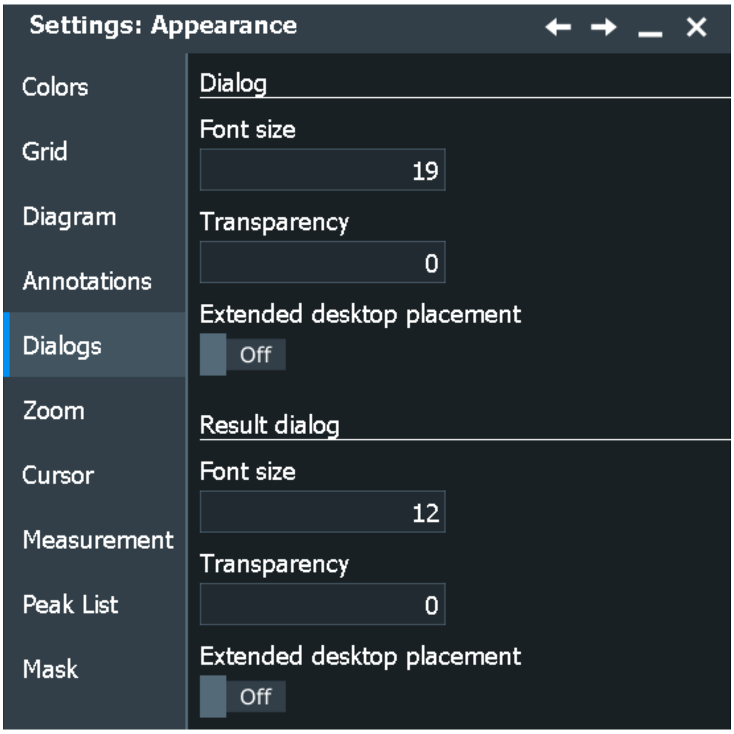
"Add"	Adds a line at the end of the list.
"Copy"	Copies the selected line in a new line.
"Remove"	Deletes the selected line. Only single lines can be removed. You can also delete a label by using the toolbar: Tap the "Delete" icon and then the label.

Remote command:

`DISPlay:SIGNal:LABel:ADD` on page 1308
`DISPlay:SIGNal:LABel:REMove` on page 1310
`DISPlay:SIGNal:LABel:TEXT` on page 1311
`DISPlay:SIGNal:LABel:HORizontal:ABSolute:POSition` on page 1312
`DISPlay:SIGNal:LABel:VERTical:ABSolute:POSition` on page 1312
`DISPlay:SIGNal:LABel:HORizontal:RELative:POSition` on page 1312
`DISPlay:SIGNal:LABel:VERTical:RELative:POSition` on page 1312

4.3.6 Dialogs appearance settings

Access: "Settings" > "Appearance" > "Dialogs" tab



Dialog

With these settings, you can change the appearance of the dialogs.

Font size ← Dialog
Defines the font size of the text in dialog boxes.

Dialog transparency ← Dialog
Defines the transparency of the dialog box background. For high transparency values, you can see the waveform display in the background, and possibly check the effect of the changed setting. For lower transparency values, readability in the dialog box improves.

Result dialog
With these settings, you can change the appearance of the result dialogs.

Font size ← Result dialog
Defines the font size of the text in result boxes. The size of the result box is adapted to the font size.

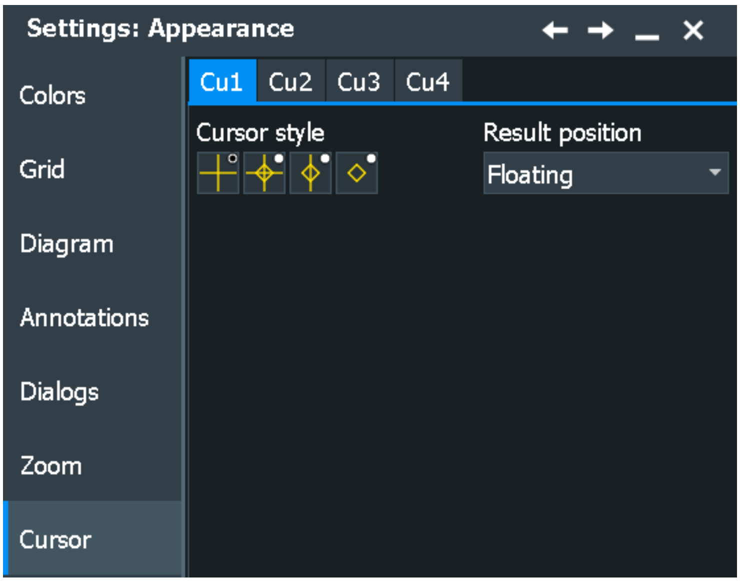
Transparency ← Result dialog
Defines the transparency of the measurement result boxes in the same way as Dialog transparency.

Extended desktop placement
If an external monitor is connected to the instrument, you can enable these settings to display dialog boxes and/or result boxes on the external monitor. Thus, the boxes do not cover the waveforms on the instrument display.

Remote command:
`DISPlay:EXTended:PORDialogs` on page 1313
`DISPlay:EXTended:POSDialogs` on page 1313

4.3.7 Cursor appearance settings

Access: "Settings" > "Appearance" > "Cursor" tab



Cursor style

Defines how the cursor is displayed in the diagram.

"Lines"	The cursors are displayed as lines.
"Line & Rhombus"	The cursors are displayed as lines. The intersections of the cursors with the waveforms are displayed by rhombus-shaped points.
"Vertical line and rhombus"	The cursors are displayed as vertical lines. The intersections of the cursors with the waveforms are displayed by rhombus-shaped points.
"Rhombus"	The intersections of the cursors with the waveforms are displayed by rhombus-shaped points.

Remote command:
`CURSor<m>:STYLe` on page 1313

Result position

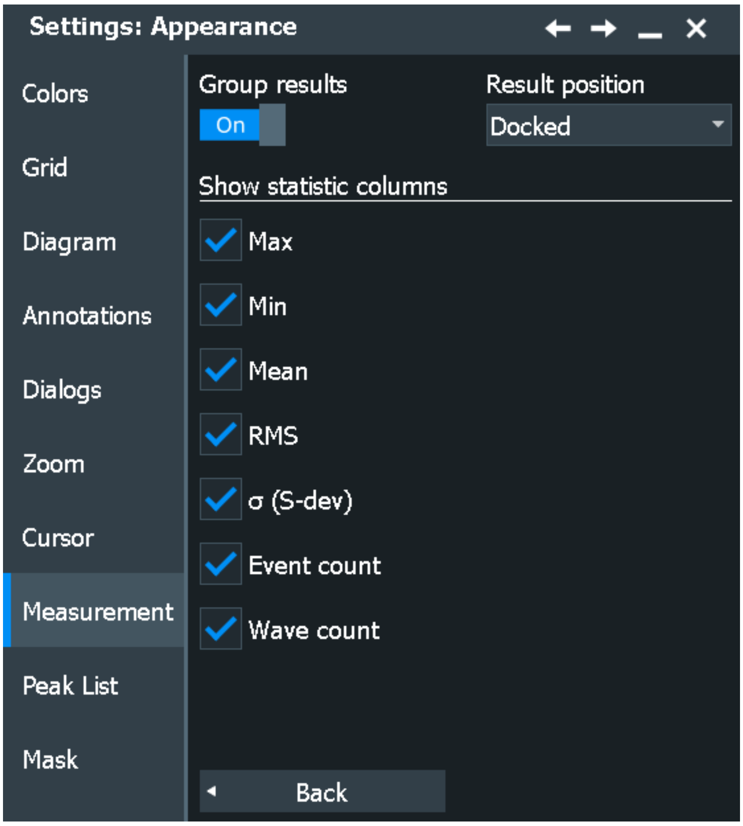
Defines the position of the cursor measurement results.

"Floating"	Floating result box in front of the diagrams.
"Preview"	Result icon on the signal bar.
"Docked"	Fixed tab below the diagrams.

Remote command:
`DISPlay:RESultboxes:CUPosition` on page 1314

4.3.8 Measurement appearance settings

Access: "Settings" > "Appearance" > "Measurement" tab



Group results

If enabled, all results are shown in one result box (default).

If disabled, the results of each measurement group are shown in a separate result box. The default position is ignored.

The setting affects all measurements except for the peak list.

Remote command:
`MEASurement<m>:DISPlay:GROuping` on page 1314

Result position

Defines the default position of the result table, when a new measurement is started:
- "Docked": fixed tab below the diagrams
- "Preview": result icon on the signal bar
- "Floating": floating result box in front of the diagrams

The setting affects only grouped results.

Remote command:
`DISPlay:RESultboxes:MEPosition` on page 1314

Show statistic columns

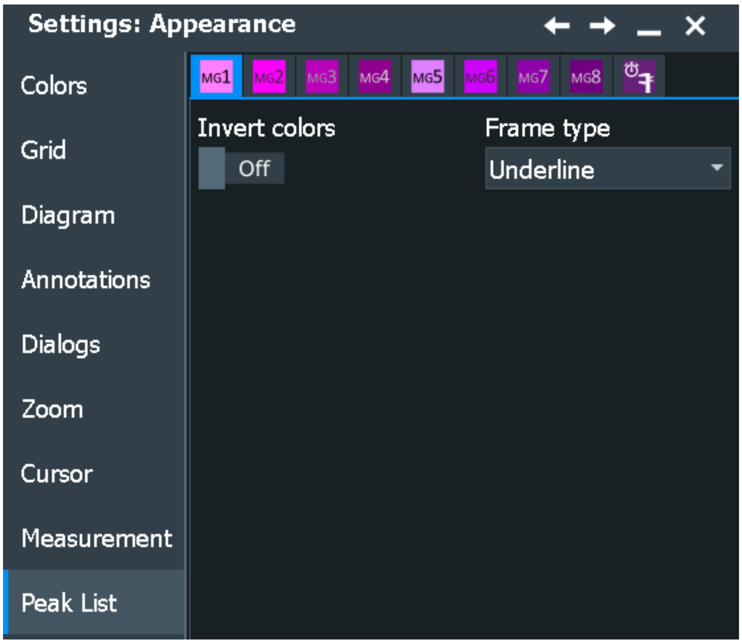
Select the statistical values that you want to see in the results table.

4.3.9 Peak list appearance settings

Access: "Settings" > "Appearance" > "Peak List" tab

For peak lists measurements only, requires option R&S RTO-K18. You can enable the "Peak list" for "Spectrum" measurements, see Chapter 8.2.7, "Spectrum measurements", on page 338.

In this dialog, you can configure the look of the labels for peaks in the spectrum diagram.



Frame type
Defines the layout of the labels (full border, underline, or none).

Remote command:
`MEASurement<m>:RESult:LABorder` on page 1315

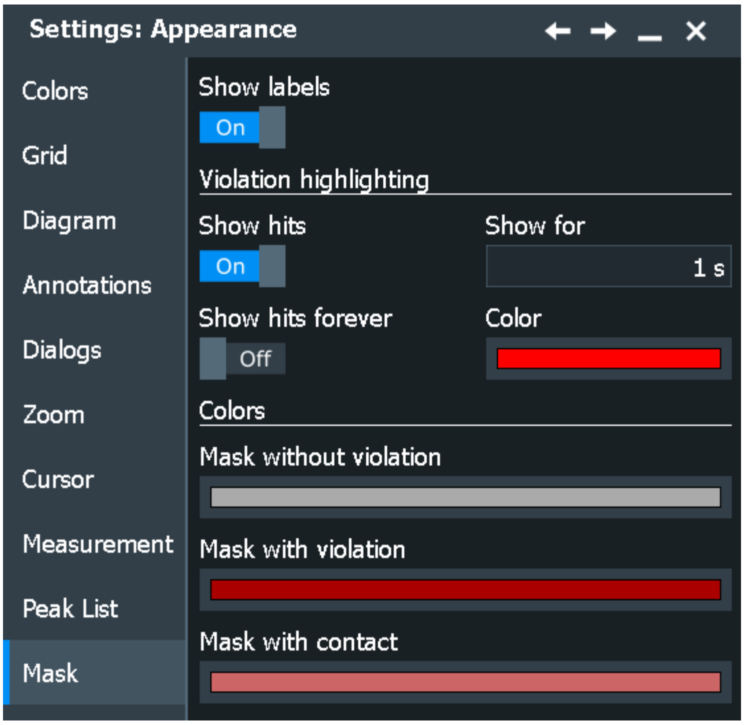
Invert
Displays black font on white background using the "Full border" frame type.

Remote command:
`MEASurement<m>:RESult:INVerse` on page 1315

4.3.10 Mask appearance settings

Access: "Settings" > "Appearance" > "Mask"

In this dialog, you can define the settings for mask and hit display. See also Chapter 10, "Mask testing", on page 393 .



Show labels
Switches the display of the mask test name on or off.

To change the name of the mask test, open the "Mask Test" dialog > "Define Mask" tab. Double-tap the mask test subtab and enter the new mask test name.

Remote command:
`MTESt:LABel` on page 1316
`MTESt:REName` on page 1316

Show hits
If selected, the mask hits are highlighted on the screen. You can define the color and the time of the hit display.

These settings are not applied to trigger zones, which are based on masks.

Remote command:
`MTESt:HIGHlight:STATe` on page 1317

Show for
Sets the time for which the mask hits are shown.

Remote command:
`MTESt:HIGHlight:TIME` on page 1317

Show hits forever
If selected, the mask hits are shown for an unlimited period of time.

Remote command:
`MTESt:HIGHlight:INFinite` on page 1316

Color

Sets the color of samples that violated the mask.

Remote command:
`MTESt:COLor:MATCh` on page 1317

Mask without violation

Sets the color of masks segments that were not hit.

Remote command:
`MTESt:COLor:UNMatch` on page 1317

Mask with violation

Sets the color of mask segments the signal has entered into.

Remote command:
`MTESt:COLor:INTerior` on page 1318

Mask with contact

Sets the color of masks segments that were touched at the border. In this case, the resolution is not sufficient to detect if the mask was hit or not. Zoom into the concerned area to see the actual result.

Remote command:
`MTESt:COLor:BORDer` on page 1318

4.3.11 Protocol appearance settings

Access: "Settings" > "Appearance" > "Protocol"

In this dialog, you can define the position of the result table for the protocol analysis.

See also Chapter 13, "Protocol analysis", on page 480.

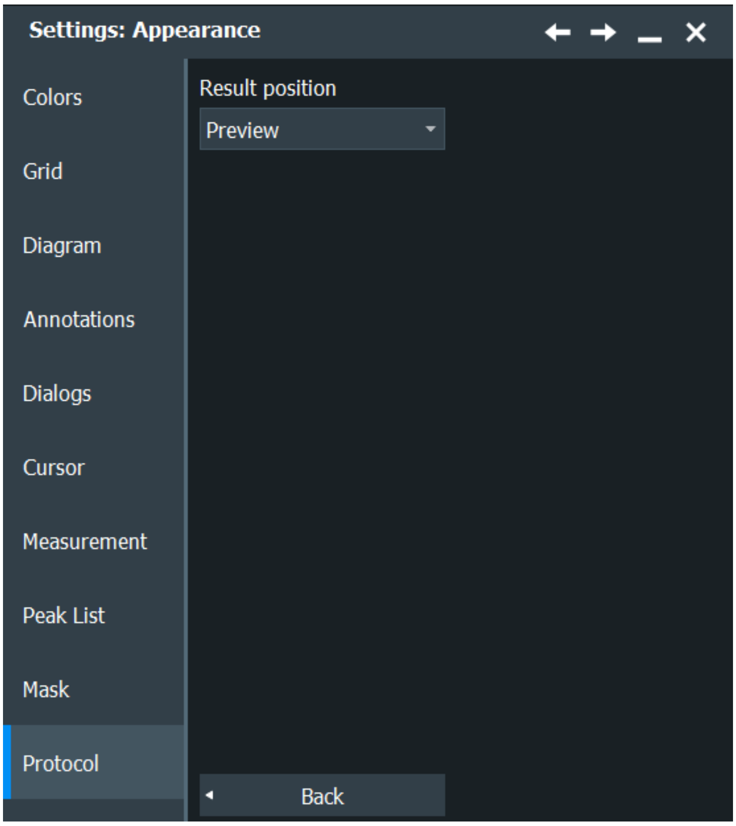


Table position

Defines the position of the decode table on the screen.

"Floating"	Floating result box in front of the diagrams.
"Preview""	Result icon on the sidebar.
"Docked"	Fixed tab below the diagrams.

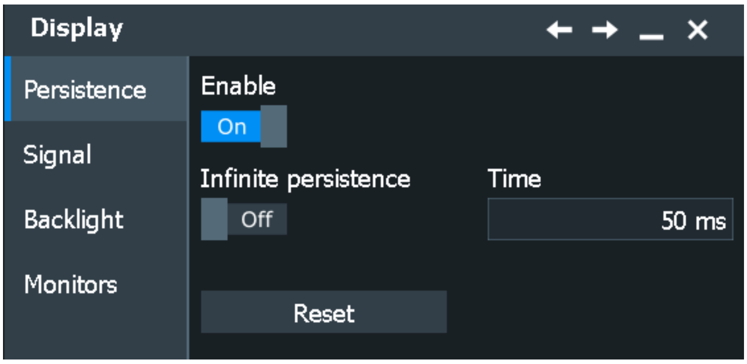
Remote command:
`DISPlay:RESultboxes:DEPosition` on page 1672

4.4 Display settings

In the "Settings" > "Display" dialog box, you can define the display settings like brightness and signal intensity. Yo can also set up an external monitor.

4.4.1 Persistence settings

Access: "Menu" > "Settings" > "Display" > "Persistence"



Enable
If enabled, each new data point in the diagram area remains on the screen for the duration that is defined using Time, or as long as Infinite persistence is selected.

If disabled, the waveform points are displayed only for the current acquisition.

Remote command:
`DISPlay:PERSistence[:STATe]` on page 1319

Infinite persistence
If infinite persistence is enabled, each new waveform point remains on the screen until this option is disabled. Use infinite persistence to display rare events in the signal.

Remote command:
`DISPlay:PERSistence:INFinite` on page 1319

Time
Sets a time factor that controls how long the waveforms points fade away from the display. Thus, the R&S RTO emulates the persistence of analog phosphor screens.

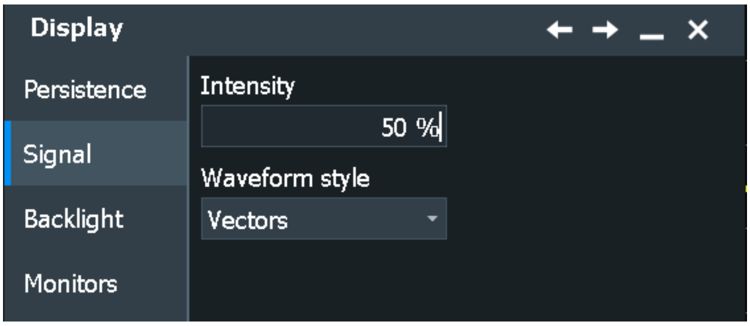
Remote command:
`DISPlay:PERSistence:TIME` on page 1319

Reset
Resets the display, removing persistent all waveform points.

Remote command:
`DISPlay:PERSistence:RESet` on page 1319

4.4.2 Signal settings

Access: "Menu" > "Settings" > "Display" > "Signal"



Intensity
This value determines the strength of the waveform line in the diagram. Enter a percentage between 0 (not visible) and 100% (strong). The default value is 50%.

You can also use the [Intensity] knob on the left side of the screen to adjust the waveform intensity directly.

Note: Use of color tables. The exact mapping of the cumulative value occurrences according to the assigned color table is guaranteed only if the intensity is set to 50%. All other intensity values falsify the mapping but can improve the visibility of the signal.

Remote command:
`DISPlay:INTensity` on page 1320

Waveform style
Select the style in which the waveform is displayed:

"Vectors" The individual waveform points are connected by a line.
Define the strength of the line using the [Intensity] knob.

"Dots" Only the individual waveform points are displayed. Waveform sample points are the ADC sample points and additional interpolated points if "Interpolated time" is used for resolution enhancement. To see the dots of one waveform, perform one acquisition with [Single] and N=1 ("Average count" = 1). During continuous acquisition, or a [Single] acquisition with N > 1, the dots of multiple subsequent waveforms are displayed on the screen, and the waveform looks like a line.
Consider also the "Interpolation" on page 133.

Remote command:
`DISPlay:DIAGram:STYLe` on page 1320

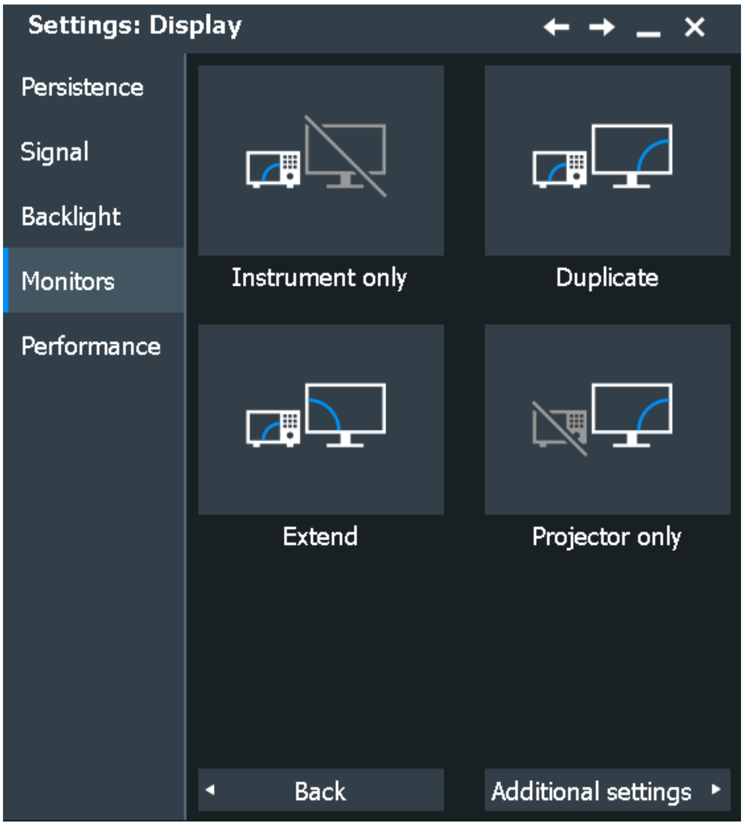
4.4.3 Backlight settings

Access: "Menu" > "Settings" > "Display" > "Backlight"

Brightness
Changes the background luminosity of the touchscreen.

4.4.4 Monitors settings

Access: "Menu" > "Settings" > "Display" > "Monitors"



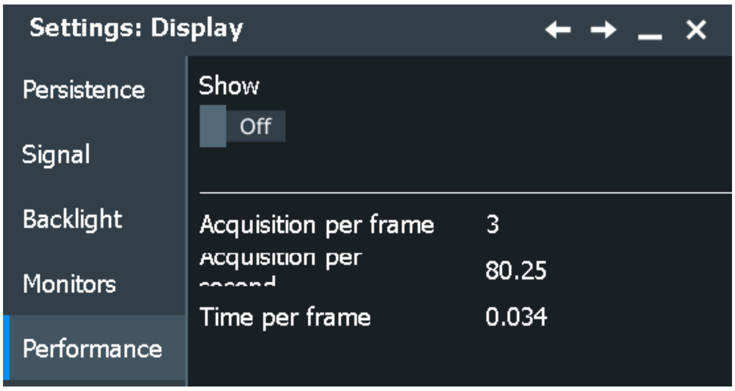
In the "Monitors" tab you can extend or duplicate the instrument display to a second monitor or projector (external display). The following options are available:

- "Instrument only": the instrument display is on, the external display is off.
- "Duplicate": the external display shows the same content as the instrument display.
- "Extend": the instrument display and the external display show different content.

- "Projector only": the instrument's user interface is only shown on the external display, the instrument display is off.
- "Additional settings": opens the Windows configuration for display settings.

4.4.5 Performance settings

Access: "Menu" > "Settings" > "Display" > "Performance"



Show

Displays the "Performance" reults box.

The "Performance" result box shows information on the current acquisition performance values of the R&S RTO.

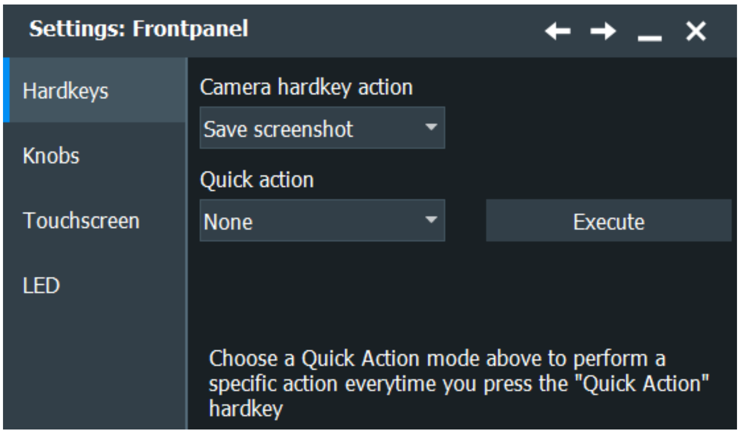
The instrument groups acquired waveforms together in a frame, and displays the frame content. The maximum number of frames displayed per second is about 30. The current number of frames per second is indicated as reciprocal "Time per frame". If the time scale decreases, and thus the number of acquisitions per second also decreases, the number of acquisitions per frame can drop to 1.

4.5 Frontpanel settings

In the "Frontpanel" dialog box, you can assign functions to keys and knobs, adjust the Navigation knob and adjust the brightness of the keys.

4.5.1 Hardkeys: function assignment

Access: "Settings" > "Frontpanel" > "Hardkeys"



Camera hardkey action

The Camera key on the left side of the display is a shortcut key that initiates an associated action.

You can assign one of the following actions:

- Save a screenshot
- Open screenshot setup
- Save a report
- Open report setup

Configure the settings for the selected action.

- Screenshots: "Save/Recall" key > "Save" tab > "Screenhot", see Chapter 12.3.1, "Screenshot settings", on page 472.
- Report: "Save/Recall" key > "Save" tab > "Report", see Chapter 12.4.1, "Report settings", on page 476.

Quick Action

The Quick Action key on the left side of the display is a shortcut key that initiates an associated action. To test the setup, tap "Execute".

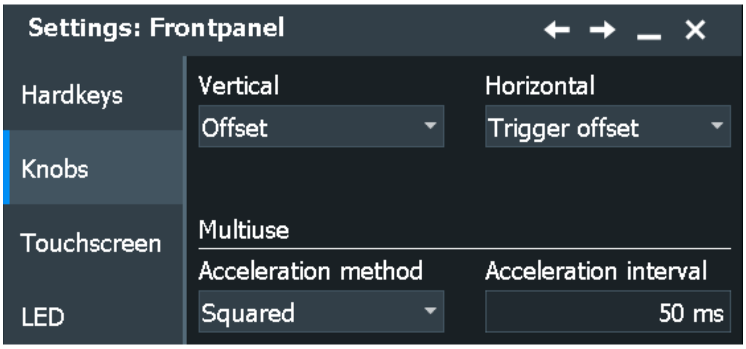
You can assign one of the following actions to the Quick Action key:

"Application"	Starts an external application. Select the path of the application executable, additional parameters, and the working directory as in a Windows shortcut definition.
"Graphical Recall"	Opens the "Load saveset" window to select and load instrument settings that were previously stored in a saveset. See also:"Graphical Saveset" on page 59 .
"Clear all"	Deletes the all measurement results, waveforms, and the history. See also: "Clear" on page 59

Remote command:
`QACTion:MODE` on page 1321
`QACTion:EXECute` on page 1321
To run an application:
`QACTion:PATH` on page 1321
`QACTion:PARameters` on page 1322
`QACTion:WDIRectory` on page 1322

4.5.2 Knobs

Access: "Settings" > "Frontpanel" > "Knobs"



Vertical
The vertical Position knob can change the waveform position or the offset of the selected waveform. Select the action that you want to perform.

See also: "[Position] (upper knob)" on page 40

Horizontal
The horizontalPosition knob can change the horizontal position or the reference point. Select the action that you want to perform.

See also:"[Position]" on page 38

Acceleration method
Selects a method to accelerate the movement of the element on the screen compared to the actual movement of the [Navigation] knob.

Acceleration is useful if you need to move from one end of the screen to the other, for example. Without acceleration, you have to turn the knob quite a while to reach the other end. On the other hand, acceleration can make precise selection difficult, since a small movement of the knob causes a relatively large movement on the screen.

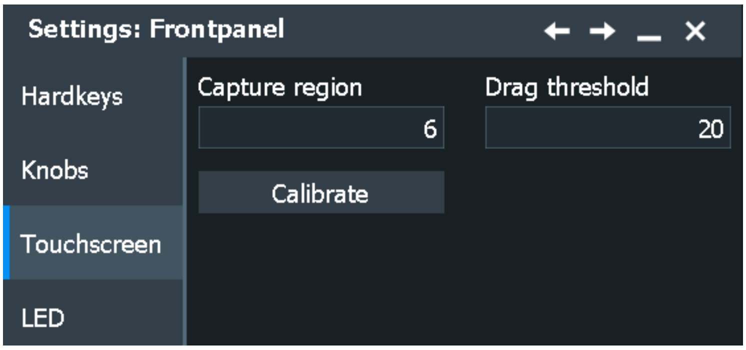
"None"	No acceleration method used.
"Squared"	Moderate acceleration method used.
"Exponential"	Strong acceleration method used.

Acceleration interval
Defines the delay time during which the movement of the [Navigation] knob is analyzed before acceleration is applied. For short intervals, acceleration sets in quickly, but is not as effective. For long intervals, acceleration is more effective. However, it takes longer until the instrument reacts on the knob's input.

Furthermore, when you turn the knob slowly during fine-tuning, subsequent movements that occur during the same interval are accelerated, making precise selection difficult.

4.5.3 Touchscreen

Access: "Settings" > "Frontpanel" > "Touchscreen".



Capture region
Defines the number of pixels around each element (e.g. button, icon, data point) that create a capture region. If you tap your finger or click the mouse pointer within this capture region, this element is considered to be selected. If you tap or click outside this area, a different or no element is selected.

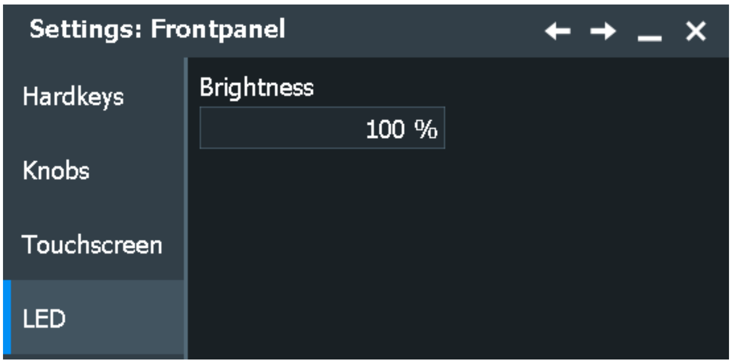
The larger the region, the easier is it to select an element. However, when selecting data points, for example, a large frame does not allow you to select precisely.

Drag threshold
Defines the maximum number of pixels around an element (e.g. data point) within which your pointing device must stay to "click" the element. When you tap or click a specific element and move your finger or the mouse outside this range, it is considered to be a "dragging" or "moving" operation.

Calibration
Opens the touchscreen calibration application. Only users with administrator rights can fulfill this task.

4.5.4 LED

Access: "Settings" > "Frontpanel" > "LED".



Brightness
Defines the luminosity of illuminated front panel keys and knobs.

4.5.5 Aligning the touchscreen

When the device is delivered, the touchscreen is initially calibrated. However, to ensure that the touchscreen responds to the finger contact correctly, a touchscreen alignment is required. Only users with administrator rights can fulfill this task.

Alignment of the touchscreen is useful:

- At first use
- If the position of the instrument has been changed, and you cannot look straight on the screen
- If another person operates the instrument
- If you notice, that touching a specific point on the screen does not achieve the correct response

1. In the "Menu" > "Settings" menu, select "Frontpanel".
2. Select the "Touchscreen" tab.
3. Tap "Calibrate".

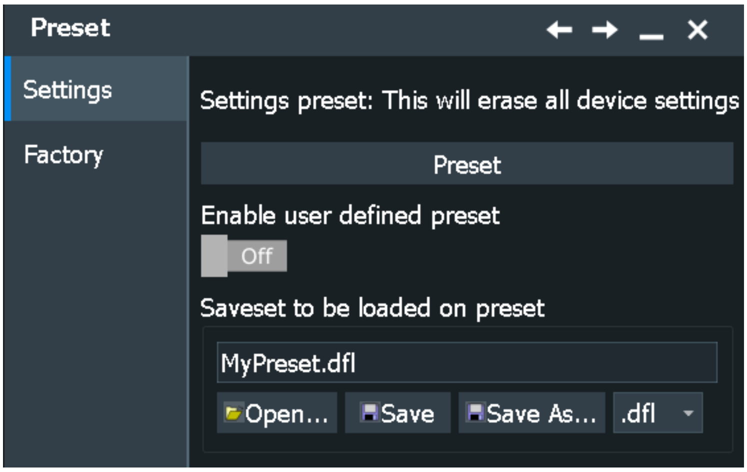
   A blinking cross appears in the lower left corner of the screen.
4. Touch and hold the cross until "OK" is shown.
5. Repeat this action for the crosses in the other corners.
6. Tap the Rohde & Schwarz logo to display the instrument's user interface.

4.6 Preset setup

A user-defined preset contains the complete instrument setup including display settings, except for transparency and intensity. You can save the current configuration to a preset file, and load a previously saved preset file. You can then specify that these settings are to be applied with the [Preset] key.

4.6.1 User-defined preset - settings

Access: "Menu" > "Settings" > "Preset" > "Settings" tab



Preset
Performs a user-defined preset of the instrument.

Saveset to be loaded on preset
The file name with extension `.dfl` to load or to save the settings to.

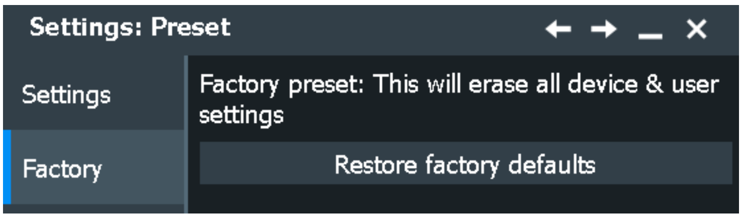
For details, see .Chapter 12.5, "File selection dialog", on page 478.

Enable user defined preset
If enabled, the settings from the selected preset file are restored when the [Preset] key is pressed.

If disabled, [Preset] sets the instrument to the factory defaults.

4.6.2 Factory preset

Access: "Menu" > "Settings" > "Preset" > "Factory"



Restore factory defaults
Resets the instrument to the factory default settings, to the initial state. Factory settings comprise all instrument settings, including display, intensity and transparency settings. After loading factory defaults, perform a self-alignment to synchronize the signal data.

Remote command:
Chapter 23.7.1, "System", on page 1297

4.6.3 Restoring settings

When you have changed many different settings on the instrument and are no longer sure which settings are causing which effect in the measurement, you may want to restore the default settings and start anew. The following methods are available:

- Saving instrument settings to a user-defined preset and restoring the instrument settings to user-defined default values
- Restoring all settings on the R&S RTO to the factory-defined values
- Restoring settings from a file (see "To load settings from a saveset file" on page 448)

For details on save/recall instrument settings and associated remote commands, see Chapter 12.1.1, "Savesets", on page 442.

To save a user-defined preset

1. Open the "Menu" > "Settings" > "Preset" tab.
2. Enter a name for the preset file. Select the file format.
3. Tap Save.

   Note: If you want to store the file in another directory than the default one, select "Save As". See also: Chapter 12.5, "File selection dialog", on page 478

To restore the instrument settings to user-defined default values

1. Open the "Menu" > "Settings" > "Preset" tab.
2. Tap "Open" and select the preset file that contains the required settings.

   The instrument settings are restored to values that are stored in the file.
3. To use these settings as preset values, select "Enable user-defined preset".
4. Press the [Preset] key.

To restore all settings to the factory defaults

1. Open the "Menu" > "Settings" > "Preset" > "Factory " tab.
2. Tap the "Restore factory defaults" button.

   All settings on the R&S RTO are reset to their factory-defined values. As long as no user-defined preset file is loaded and "Enable user defined preset " is disabled, the [Preset] key also resets the instrument settings to factory defaults.

4.7 Maintenance settings

In the "Settings" > "Maintenance" dialog box, you define the the look and feel of the display element, e.g. waveform colors, result position, or grid behavior.

4.7.1 Firmware update

Access: "Menu" > "Settings" >"Maintenance" > "FW Update"

Your instrument is delivered with the latest firmware version. Firmware updates are provided on the internet at:

www.rohde-schwarz.com/firmware/rto.

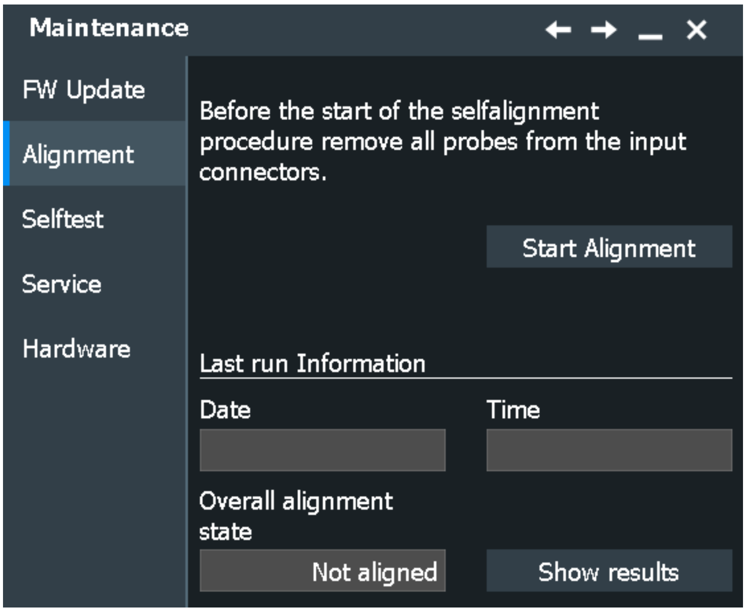
The "Release Notes" describe the improvements and modifications of all firmware versions. They also explain how to update the firmware. They are available along with the firmware on the same web page.

4.7.2 Alignment

When data from several input channels is displayed at the same time, it may be necessary to align the data vertically or horizontally to synchronize the time bases or amplitudes and positions. This is the case, for example, when strong temperature changes occur (> 5°).

4.7.2.1 Control

Access: "Menu" > "Settings" >"Maintenance" > "Alignment"



Start Alignment
Starts the self-alignment procedure for all channels.

Remote command:
*CAL? on page 1288

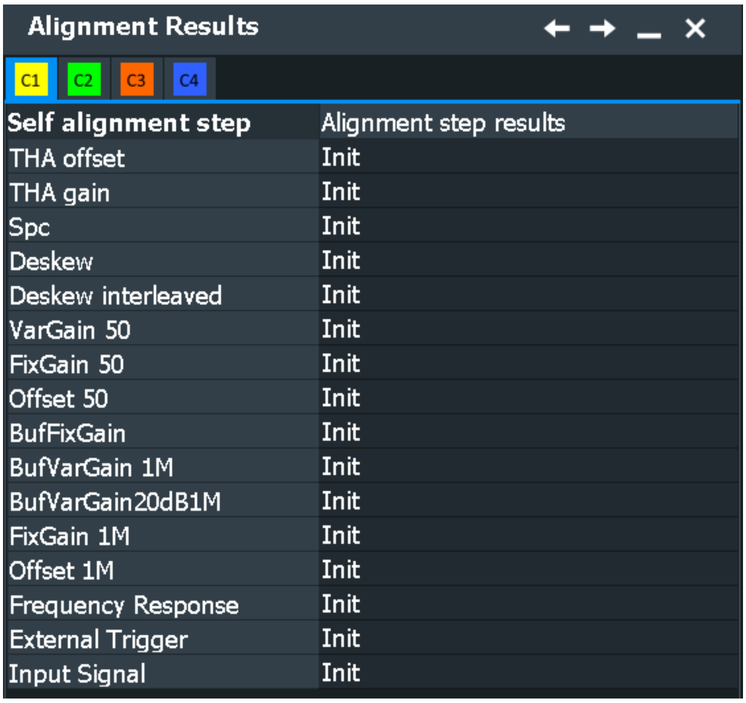
Date / Time / Overall alignment state
Show the date, time and the summary result of the self-alignment process: not aligned, passed or failed. Detailed results are provided on the "Results" tab.

Remote command:
`CALibration:DATE?` on page 1323
`CALibration:TIME?` on page 1324
`CALibration:RESult?` on page 1324

4.7.2.2 Results

For each channel, the results of the individual alignment steps are shown for all technical channel component. In case you require support, you may be asked to provide this information.

Access: "Settings" >"Maintenance" > "Alignment" > "Show results"



4.7.2.3 Performing a self-alignment

The self-alignment aligns the data from several input channels vertically and horizontally to synchronize the timebases, amplitudes and positions. The self-alignment process includes a basic hardware check.

Recommendation on performing the self-alignment:

- When putting the instrument into operation for the first time
- After a firmware update
- Once a week
- When major temperature changes occur (> 5°)

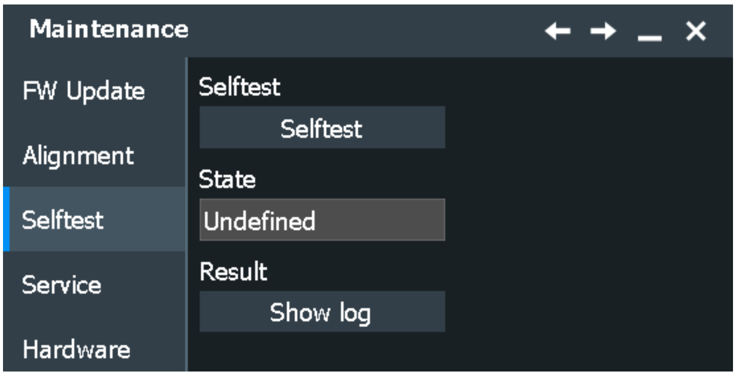
1. Warm up the instrument before you start the self-alignment. The minimum warm-up time is indicated in the data sheet.
2. Remove the probes from the input connectors.
3. In the "Menu" > "Settings" menu, select "Maintenance".
4. In the "Alignment" tab, tap "Start Alignment".

   The alignment is performed, the process might take several minutes. A message box informs you about the running process, wait until this message box closes. The overall pass/fail result is shown in the "Overall alignment state" field. The results of the individual alignment steps for each input channel are indicated in the "Results" tab. This information is required if problems arise.

4.7.3 Self-test

The instrument's self-test checks the hardware for correct operation. Perform the self-test if you suspect problems in hardware operation.

Access: "Settings" >"Maintenance" > "Selftest"



The test can take several minutes. The summary result is shown in the "State" field, which can be helpful in case you need support.

Selftest
Starts the self-test.

Remote command:
*TST? on page 1292

State
Shows the summary result of the self-test: Pass or Fail.

Remote command:
DIAGnostic:SERVice:STST:STATe? on page 1324

Result
Opens a log file with detailed information on the self-test steps and operation of hardware components, which can be helpful in case you need support.

4.7.4 Maintenance information

Access: "Settings" >"Maintenance" > "Hardware" tab > "Advanced"

The "Maintenance" dialog box provides information on your R&S RTO configuration, which can be helpful in case you need support.

System Info
This tab provides general information on the hardware configuration, and indicates where system information can be found on the instrument. Here you can also show the content of the device footprint file.

Remote command:
`DIAGnostic:SERVice:PARTnumber` on page 1325
`DIAGnostic:SERVice:SERialnumber?` on page 1325
`SYSTem:DFPRint` on page 1324

Mainboard
This tab provides information on the mainboard configuration in your instrument.

Frontend
This tab provides information on the frontend configuration in your instrument.

Frontpanel
This tab provides information on the front panel module installed in your instrument.

MSO
This tab is only relevant if the MSO option R&S RTO-B1 is installed. The tab provides information on the MSO hardware module that is installed in your instrument.

AWG
This tab is only relevant if the waveform generator option R&S RTO-B6 is installed. The tab provides information on the generator hardware module that is installed in your instrument.

PSC
This tab is only relevant if the pulse source option R&S RTO-B7 is installed. The tab provides information on the pulse source hardware module that is installed in your instrument.

Service
This tab allows the service personnel to enter a password that activates further service functions.

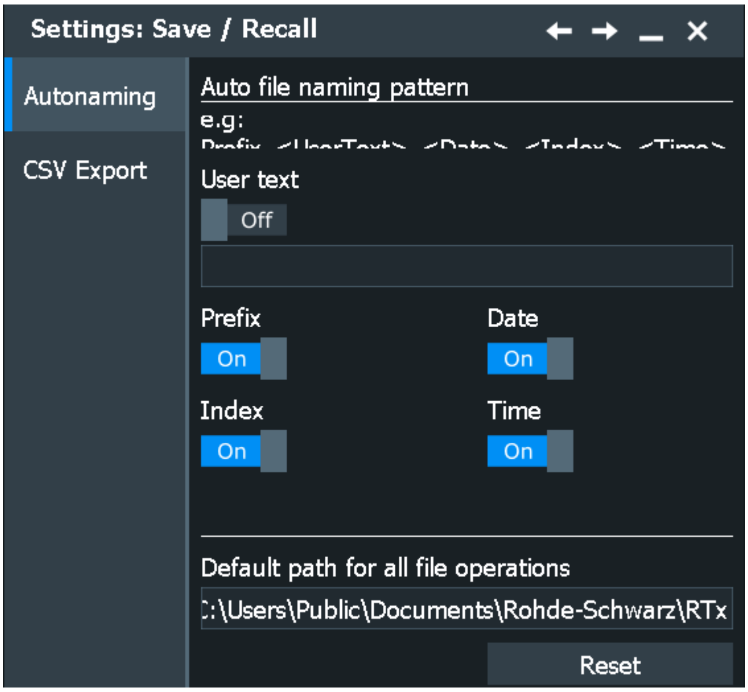
Remote command:
- `DIAGnostic:SERVice:PWD` on page 2565

4.8 Save / recall

4.8.1 Autonaming

Access:"Menu" > "Settings" key > "Maintenance" > "Autonaming" tab.

4.8.1.1 Autonaming settings



In this tab, you can define the pattern for automatic file name generation. This name is used as the default file name. The default path is the storage location for all saved files and their subdirectories.

Prefix

If enabled, inserts the default prefix in the file name. The prefix indicates the type of data that is saved, for example, Histogram, RefCurve, Settings.

Remote command:
`MMEMory:AUTonaming:PREFix` on page 1322

User text (enable)

If enabled, inserts the specified user text after the prefix.

Remote command:
`MMEMory:AUTonaming:USERtext` on page 1322

Text input

User-defined text to be inserted after the prefix.

Remote command:
MMEMory:AUTonaming:TEXT on page 1323

Date
If enabled, inserts the current date.

Remote command:
MMEMory:AUTonaming:DATE on page 1322

Index
If enabled, inserts an index.

Remote command:
MMEMory:AUTonaming:INDex on page 1322

Time
If enabled, inserts the current time.

Remote command:
MMEMory:AUTonaming:TIME on page 1322

Default path for all file operations
Defines the default path displayed in the file selection dialog box for loading and storing operations. If a USB flash drive is connected, the path is set automatically to the drive letter of the USB flash drive.

To switch the directory quickly, double-tap the input field. Use the symbols on the left of the file explorer box to change the directory.

Remote command:
MMEMory:AUTonaming:DEFaultpath on page 1323

Reset
Resets the default file path to the factory default.

Remote command:
MMEMory:AUTonaming:RESPath on page 1323
MMEMory:AUTonaming:RESall on page 1323

4.8.1.2 Defining default file paths and names

When a save or load operation is performed, a default file name and path is provided. You can configure which path is used and how the file name is generated. In the file selection dialog box, you can change the folder and name as desired.

To define the default file path

1. Tap "Menu" > "Settings".
2. Select the "Save /Recall" tab.
3. Select the "Autonaming" tab.
4. Double-tap the "Default path for all file operations" field.

The directory selection dialog box is opened.

5. Select the folder in which the data is stored by default. Use the symbols on the left of the file explorer box to switch to often used directories.
6. To restore the factory-set default path, tap "Reset" next to the path field.

To define the automatic file name pattern

The automatic file name pattern can consist of the following elements:

<Prefix>_<UserText>_<Date>_<Index>_<Time>

The prefix depends on the data type to be stored and cannot be changed by the user. The other elements can be enabled or disabled as required.

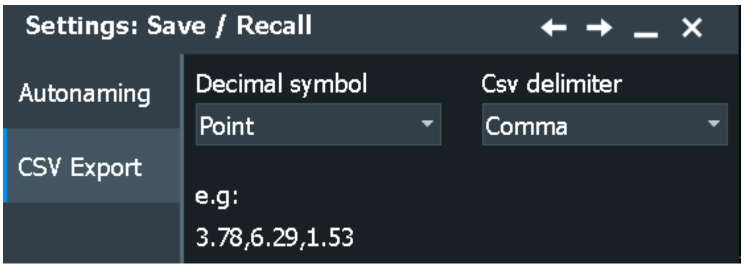
1. Tap "Menu" > "Settings".
2. Select the "Save /Recall" tab.
3. Select the "Autonaming" tab.
4. To insert a user-defined text after the prefix, enter the text in the edit field. and enable "User text".
5. If you want to exclude the prefix, current date, time or an index (serial number), disable the corresponding option.

   The specified elements are used to generate the default file name for the next storage operation.

4.8.2 CSV export

Access: "Menu" > "Settings" key > "Save /Recall" > "CSV Export" tab.

In this dialog, you can define the format of your CSV file.



Decimal symbol
Selects if point or comma is used as a decimal symbol in the exported CSV file.

Remote command:
EXPort:RESult:DECSymbol on page 1667

Csv delimiter
Selects the list separator symbol from a list. Available are semicolon, comma, space, tab and colon.

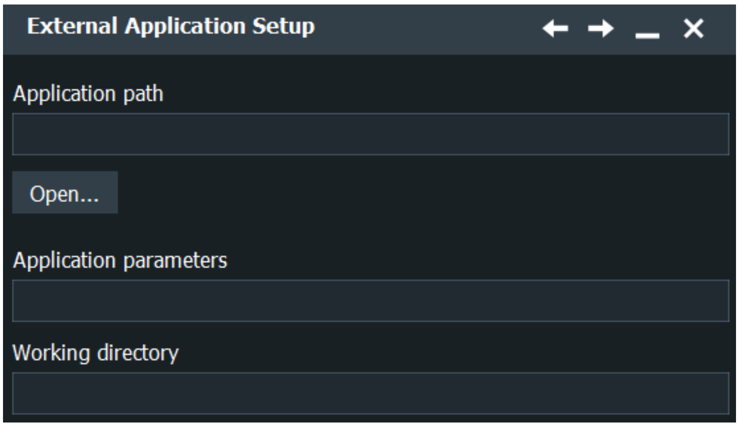
Remote command:
EXPort:RESult:DELimiter on page 1667

4.8.3 External application

The R&S RTO can start an external application on the instrument or in the network (if connected) when an event occurs.

The following events can start an application:

- Trigger event
- Mask violation
- Successful completion of mask test
- Limit or margin violation of measurements
- Successful completion of limit and margin tests



Set the path of the application executable, optional parameters, and the working directory as in a Windows shortcut definition. The setup is valid for all events.

Remote commands:

- EXECutable:NAME on page 1667
- EXECutable:PARameter on page 1667
- EXECutable:WDIRectory on page 1668

5 Acquisition and waveform setup

This chapter describes the horizontal and vertical settings as well as the acquisition and probe setup.

5.1 Basics

This chapter provides background information on the essential settings in the vertical and horizontal systems, on acquisition setup and probing.

5.1.1 Vertical system

The controls and parameters of the vertical system are used to scale and position the waveform vertically.

5.1.1.1 Input coupling

The input coupling influences the signal path between input connector and the following internal signal stage. The coupling can be set to DC, AC, or ground.

- DC coupling shows all parts of an input signal. DC coupling is available with 1 MΩ input impedance to connect standard passive probes. DC 50 Ω coupling is the default for 50 Ω input impedance to connect, for example, active probes.
- AC coupling is useful if the DC component of a signal is of no interest. AC coupling blocks the DC component of the signal so that the waveform is centered on zero volts.
- Ground coupling disconnects the input signal from the vertical system to see the ground level (zero volts) on the screen. Ground coupling is useful for reference purposes.

5.1.1.2 Vertical scale and position

Vertical scale and vertical position directly affect the resolution of the waveform amplitude. The vertical scale corresponds to the ADC input range. To get the full resolution of the ADC, set up the waveforms to cover most of the height of the diagram.

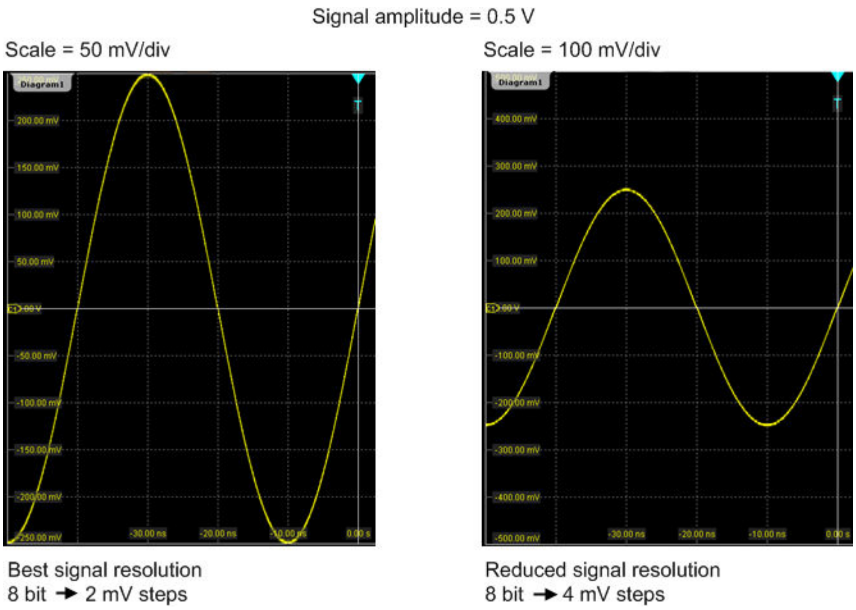


Figure 5-1: Input range and resolution of the ADC

With R&S RTO, you can work with multiple diagrams, and each diagram obtains the full vertical resolution, no matter where the diagram is placed. Therefore, use a separate diagram for each waveform instead of the traditional setup that arranges the waveforms side by side in one diagram.

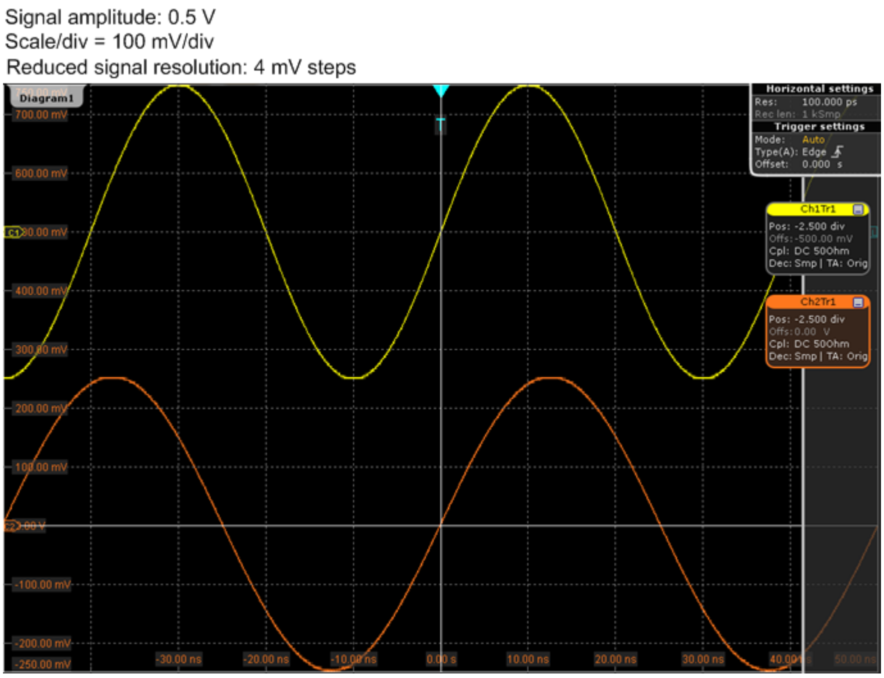


Figure 5-2: Traditional setup of multiple waveforms in one diagram: reduced resolution

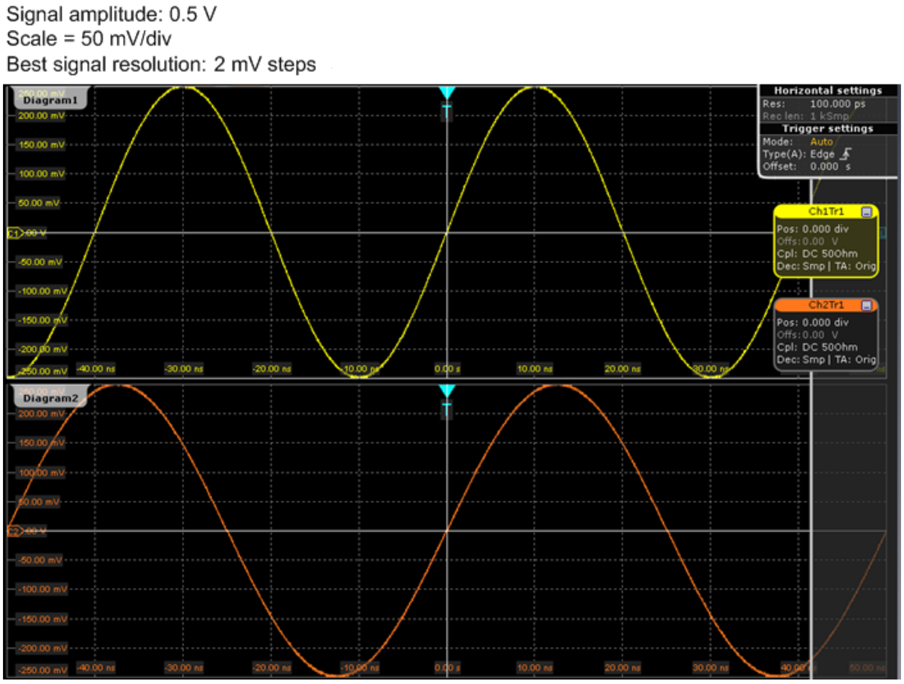


Figure 5-3: R&S RTO setup of multiple waveforms in separate diagrams: best resolution

5.1.1.3 Bandwidth

For analog applications, the highest signal frequency determines the required oscilloscope bandwidth. The oscilloscope bandwidth should be slightly higher than the maximum frequency included in the analog test signal to measure the amplitude with very little measurement error.

Most test signals are more complex than a simple sine wave and include several spectral components. A digital signal, for example, is built up of several odd harmonics. As a rule, for digital signals the oscilloscope bandwidth should be 5 times higher than the clock frequency to be measured.

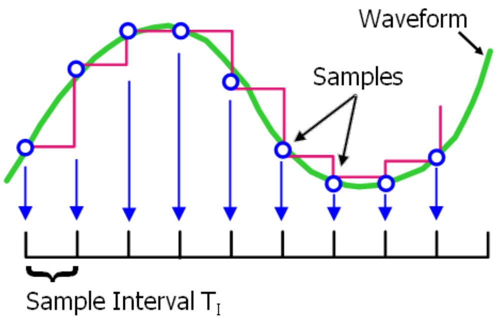
The oscilloscope is not a stand-alone system. You need a probe to measure the signal of interest, and the probe has a limited bandwidth, too. The combination of oscilloscope and probe creates a *system bandwidth*. To reduce the effect of the probe on the system bandwidth, the probe bandwidth must exceed the bandwidth of the oscilloscope, the recommended factor is 1.5 x oscilloscope bandwidth. See also: Chapter 5.1.4.1, "Voltage probes", on page 122

5.1.2 Sampling and acquisition

The vertical system of a digital oscilloscope conditions the test signal in a way that the following A/D converter (ADC) can transform the measured voltage into digital data.

5.1.2.1 Sampling and processing

The A/D converter samples the continuous signal under test at specific points in time and delivers digital values called **ADC samples**. The rate at which the converter is working is the **ADC sample rate**, a constant value specified in GHz: $f_{ADC} = 1 / T_I$



The digital ADC samples are processed according to the acquisition settings. The result is a waveform record that contains **waveform samples** and is stored in the **waveform memory**. The waveform samples are displayed on the screen and build up the waveform.

The number of waveform samples in one waveform record is called **record length**. The rate of recording waveform samples - the number of waveform samples per second - is the **sample rate**. The higher the sample rate, the better the resolution is and the more details of the waveform are visible.

Maximum sample rate on R&S RTO1044/2044/2064

R&S RTO1044/2044/2064 can work with double maximum real-time sample rate compared to other R&S RTO instruments. This high sample rate is achieved by interleaving two channels: channels 1 and 2 are interleaved, and also channel 3 and 4. Interleaving assumes that only one of the paired channels can be used - either channel 1 or channel 2, and either channel 3 or 4. If the second channel of a pair is used (on display, or as trigger source, math source, or measurement source), the interleaving mode is disabled. The real-time sample rate is limited to the usual value of 10 GSa/s. Interleaved mode is also disabled in IQ mode, if NFC or TV trigger is selected, and for most serial protocols.

Preset sets the R&S RTO1044/2044 to 10 GSa/s (non-interleaved mode), while the R&S RTO2064 is set to 20 GSa/s (interleaved mode).

Minimum sample rate and aliasing

A sufficient resolution is essential for correct reconstruction of the waveform. If the signal is undersampled, aliasing occurs - a false waveform is displayed. To avoid aliasing and accurately reconstruct a signal, Nyquist theorem postulates that the sample rate must be at least twice as fast as the highest frequency component of the signal. However, the theorem assumes ideal conditions, so the Nyquist sample rate is usually not sufficient.

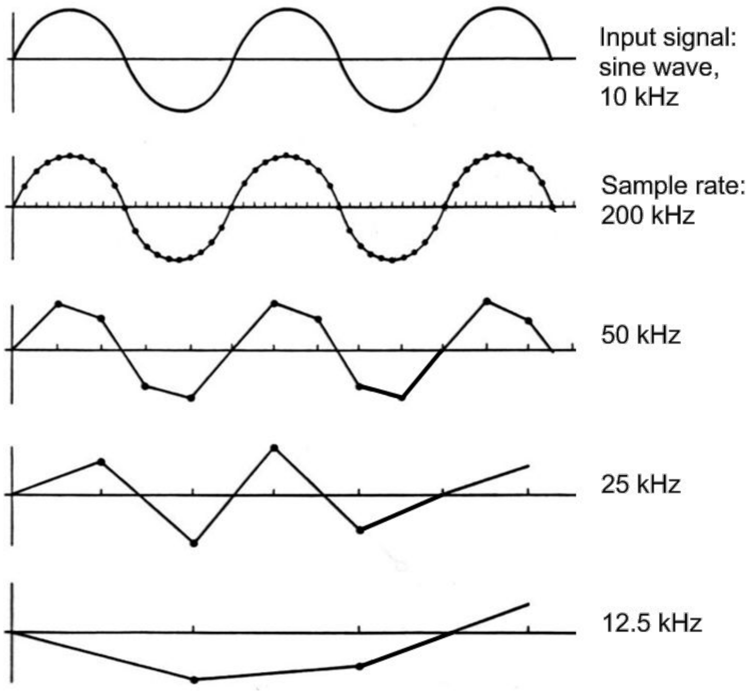


Figure 5-4: Waveforms acquired with different sample rates

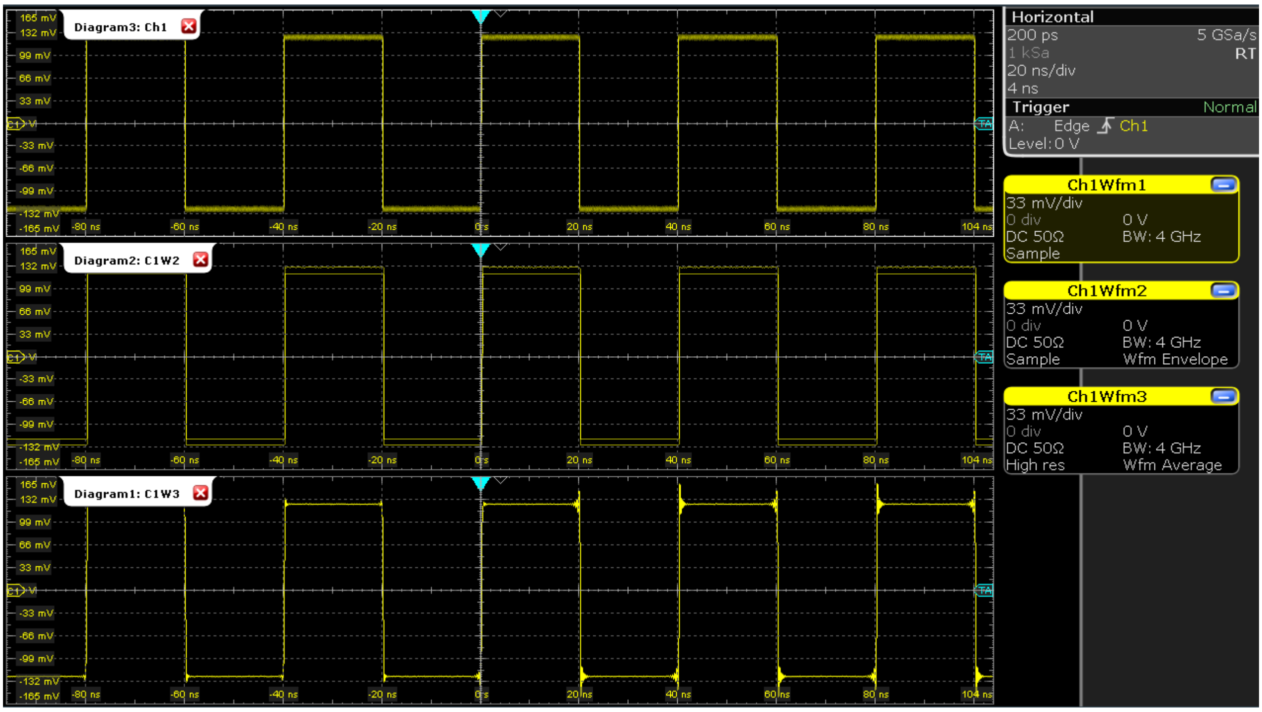
To avoid aliasing, the sample rate must be set to a value 3 to 5 times the fastest frequency component of the signal. A higher sample rate increases signal fidelity, increases the chance to capture glitches and other signal anomalies, and improves the zoom-in capabilities.

5.1.2.2 Acquisition settings

The sample rate can be the same as the constant ADC sample rate, or higher, or lower. To get a higher sample rate, interpolation as method of **resolution enhancement** is used. To reduce the sample rate, **decimation** methods help: sample, peak detect, high resolution and RMS.

As digital waveform data is stored in the memory, and the memory can save many waveform records, further **waveform arithmetic** processing is possible: average and envelope waveforms are resulting waveforms, created from a composite of sample points taken from multiple acquisitions.

You can display up to three waveforms from one input signal and apply different decimation and arithmetic to each waveform.



5.1.2.3 Acquisition control

You can run the R&S RTO in two ways:

- Run Stop: the instrument acquires data until you stop it manually.
- Single: the instrument samples and processes a specified number of acquisitions.

The determining point of an acquisition is the trigger. The instrument acquires continuously and keeps the sample points to fill the pre-trigger part of the waveform record. When the trigger occurs, the instrument continues acquisition until the post-trigger part of the waveform record is filled. Then it stops acquiring and waits for the next trigger. When a trigger is recognized, the instrument does not accept another trigger until the acquisition is complete.

The trigger modes define how the instrument triggers:

- Normal: The instrument acquires a waveform only if a real trigger occurs, that is, if all trigger conditions are fulfilled.
- Auto: The instrument triggers repeatedly after a fixed time interval if the trigger conditions are not fulfilled. If a real trigger occurs, it takes precedence. If the real trigger is faster than the auto trigger, both modes are virtually the same.

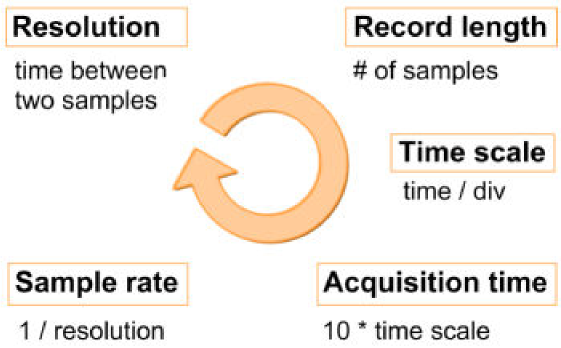
In practice, both trigger modes are useful: The auto mode lets you see the signal with little adjustment, while the normal mode selects the interesting part of the waveform. If you want to acquire a specified number of waveforms, make sure to select the normal trigger mode. Thus you get only the required number of interesting acquisitions.

See also: Chapter 6, "Triggers", on page 194.

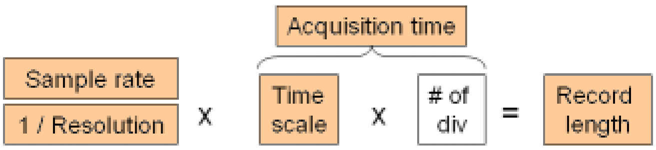
5.1.3 Horizontal system

5.1.3.1 Parameters of the horizontal system

The control parameters of the horizontal system are tightly connected. Thus, changing one parameter affects the other parameters as well.



The mathematical dependencies can be summarized as follows:



The number of divisions is 10, which is the only constant parameter.

When you set up horizontal parameters, you can choose whether the record length or the resolution remains constant.

- With constant resolution, increasing the time scale also increases the record length, and vice versa. You can limit the record length to a maximum value.
- With constant record length, increasing the time scale coarsens the resolution, that is, the time between two waveform samples gets longer.

For both settings, the "Auto adjustment" ensures a sufficient resolution to prevent undersampling.

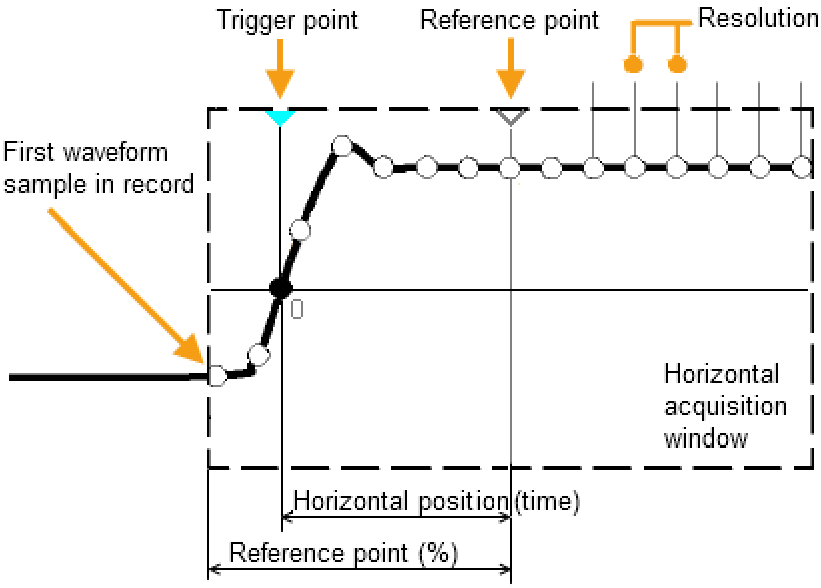
5.1.3.2 Horizontal position

As described before in Chapter 5.1.2.3, "Acquisition control", on page 119, the trigger is the determining point of the waveform record.

In many scenarios, you want to analyze the waveform some time before or after the trigger. To adjust the horizontal acquisition window to the waveform section of interest, you can use the following parameters:

- The **horizontal position** defines the time distance from the trigger point (the zero point of the diagram) to the reference point. Changing the horizontal position, you can move the trigger point, even outside the screen.

- The **reference point** is the rescaling center of the time scale on the screen. If you modify the time scale, the reference point remains fixed on the screen, and the scale is stretched or compressed to both sides of the reference point.



5.1.4 Probes

A probe connects the signal source (DUT) to the oscilloscope, and delivers the signal to be measured. It is the essential first link in the measurement chain.

An ideal probe fulfills the following requirements:

- Safe and reliable contacts
- Infinite bandwidth
- The probe should not load the signal source and thus impact the circuit operation.
- The connection should not introduce or suppress signal components (hum, noise, filter) and thus degrade or distort the transferred signal.

In reality, the probe can never be an ideal one, it always affects the signal transmission and the signal source, and thus the measured signal. It depends on the frequency to be measured and on the signal source to determine the acceptable loading, and to determine which kind of probe delivers good results.

The solution depends on the quantity to be measured regarding:

- Signal type: voltage, current, power, pressure, optical, etc.
- Signal amplitude: The oscilloscope itself can only display voltages in a limited range. Most probes can adjust the dynamic range to amplitudes from a few mV to 10 V. Smaller or much larger signals require specialized equipment.
- Signal frequency: High frequencies require advanced equipment to get correct results.

- Source characteristic: The source impedance is the decisive factor when choosing the suitable connection.

5.1.4.1 Voltage probes

The following table provides an overview on common voltage probes and their usage.

Table 5-1: Voltage probes overview

Probe type	Attenuation	Typical bandwidth range	Oscilloscope input	Usage
Passive, high impedance	1:1	10 MHz	1 MΩ	Low-speed signals, low-level signals
Passive, high impedance	10:1	500 MHz	1 MΩ	General purpose
Passive, low impedance	10:1	up to 10 GHz	50 Ω	High frequency
Active, single-ended	10:1	up to 10 GHz	50 Ω	High speed
Active, differential	10:1		50 Ω	Floating

For a list of recommended probes, refer to the R&S RTO product brochure.

Besides the possible input voltage range, two factors are important when selecting a voltage probe: Bandwidth and impedance over frequency.

- **Bandwidth:**
  The combination of probe and oscilloscope builds up a system. The resulting system bandwidth is approximately determined with:

  $$\frac{1}{BW_{system}} = \sqrt{\left(\frac{1}{BW_{probe}}\right)^2 + \left(\frac{1}{BW_{scope}}\right)^2}$$

  To measure the signal with low measurement error, the system bandwidth should be higher than the highest frequency component of the signal. The probe bandwidth must be even higher than the system bandwidth.
- **Impedance:**
  A minimum impedance is required to keep the circuit loading low. Over frequency, the impedance decreases, in particular with passive probes. The probe impedance should be approximately 10 times the impedance of the circuit test point at the highest signal frequency.

Passive voltage probes

Passive probes have the following qualities:

- No active components inside
- BNC connector for universal use
- Compensate the probe when it is connected to a scope input: LF compensation matches the probe (mainly cable) capacitance to the oscilloscope input capacitance.

- With high impedance probes, the impedance varies significantly over frequency.
- With low impedance probes, the impedance variation over frequency is low, but the load on the source is high.

If you use passive probes, remember some recommendations:

- Use a probe recommended for your oscilloscope model.
- Use a ground lead as short as possible to minimize the effect of ground lead inductance. The resonance frequency can be much lower than the system bandwidth and thus can affect the measurement results, in particular, if you measure steep edge rise times.
- Select a probe that has a bandwidth of 5 to 10 times the highest frequency being measured. This bandwidth preserves the harmonics and thus the waveform integrity.

Active voltage probes - general

Active probes require operating power from the instrument and have a proprietary interface to the instrument. Their main qualities are:

- Low loading on signal source
- The probe is automatically recognized by the instrument, no adjustment is required.
- Adjustable DC offset at probe tip allows for high resolution on small AC signals which are superimposed on DC levels.
- Connections should be as short as possible to keep the usable bandwidth high.
- Observe the operating voltage range.
- The probe impedance depends on the signal frequency.

RT-ZS single-ended active probes and RT-ZD differential active probes provide special features for easier use and precise measurements. These special features are not available on RT-ZSxxE probes.

- The micro button on the probe head remotely controls important functions on the instrument, like running and stopping the acquisition, autoset, AutoZero and setting the offset to mean value.
- The R&S ProbeMeter measures DC voltages between the probe tip and the ground connection with very high precision. The result is displayed on the instrument's screen. So you can check DC voltages with different levels without having to adjust the measurement range of the oscilloscope. The R&S ProbeMeter also measures the zero error of the probe to optimize measurement results at small signal levels.

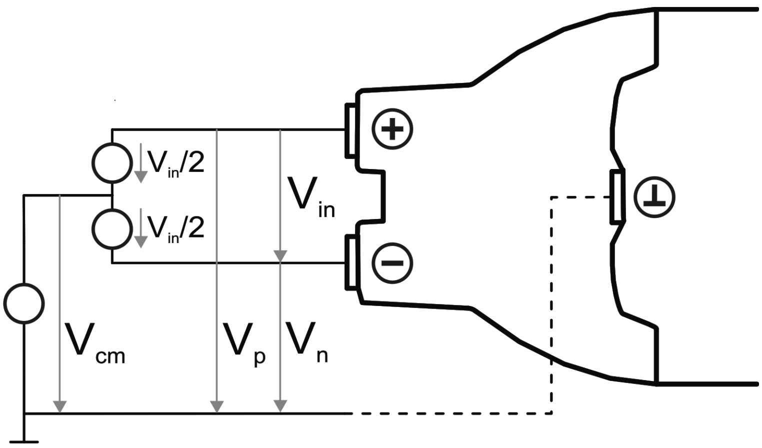
When you connect an R&S RT-ZSxx active probe to a channel input of the R&S RTO, the oscilloscope recognizes the probe. It reads the identification and calibration data from the probe box and shows the result in the "Setup" and "Probe Attributes" tabs. This data together with the deskew time for a given channel is stored and processed by the R&S RTO. If you connect the probe the next time to the same channel, the information is fetched and used.

Differential active probes

Differential active probes are designed to measure signals that are referenced against each other, and voltages that are not references to ground, for example twisted-pair signal lines. The R&S RT-ZD probes are differential probes with high input impedance, they can be used to measure voltages between any two test points.

Compared with two-channel measurement setup with single-ended probes, the measurement with differential probes is symmetric due to the same amplification and cable length on both paths. It is also immune to interference and noise and occupies only one input channel.

A differential probe has three sockets: the positive signal socket (+), the negative signal socket (-), and the ground socket.



Multiple input voltages can be defined for a differential probe:

- Differential mode input voltage (V_{in}, V_{dm})
  Voltage between the positive and negative signal sockets
- Positive single-ended input voltage (V_p)
  Voltage between the positive signal socket and the ground socket
- Negative single-ended input voltage (V_n)
  Voltage between the negative signal socket and the ground socket
- Common mode input voltage (V_{cm})
  Mean voltage of positive and negative signal sockets referred to the ground socket, respectively

Two of these voltages are independent values, the other two can be calculated:

$$V_{in} = V_p - V_n$$

$$V_{cm} = \frac{V_p + V_n}{2}$$

R&S RT-ZD probes detect only differential input voltages and provide it to the oscilloscope. Common mode signals are suppressed by the probe. This characteristic is described by the common mode rejection ratio (CMRR):

$$CMRR = \frac{DifferentialGain}{CommonMode\,Gain}$$

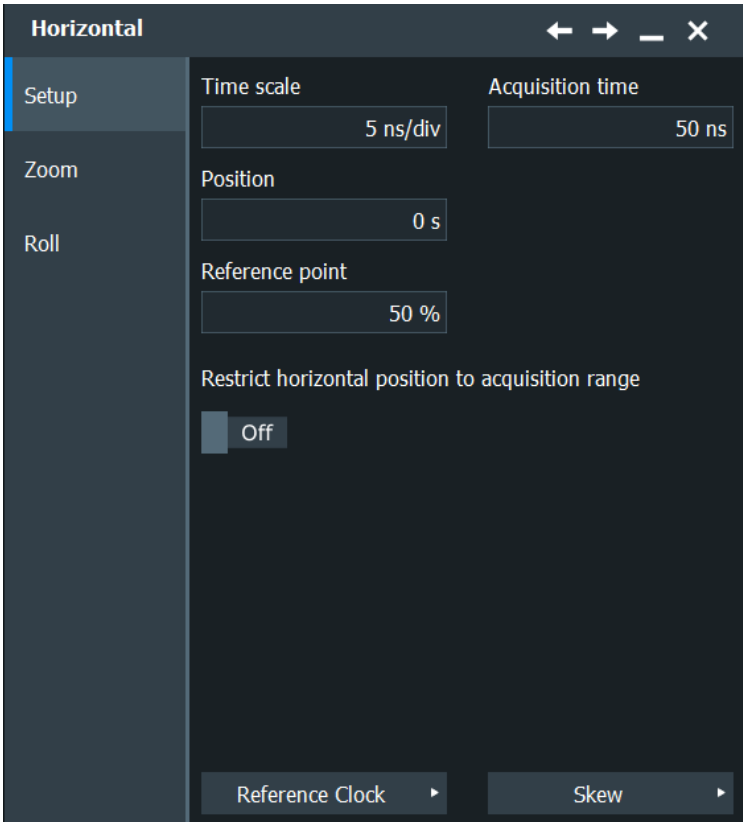
In addition, the R&S ProbeMeter of R&S RT-ZD differential probes can measure differential and common mode DC voltages. The measurement result is displayed on the oscilloscope's screen. The common mode measurement of the R&S ProbeMeter allows you to check the input voltage relative to ground. Thus, the CM measurement is a convenient way to detect breaches of the operating voltage window, and the reason of unwanted clippings.

5.2 Horizontal

The "Horizontal" dialog provides the time base configuration for channel and spectrum waveforms.

5.2.1 Setup settings

Access: "Menu" > "Horizontal" > "Setup" tab.



Time scale
Sets the horizontal scale for all channel and math waveforms in seconds per division. Increase the scale to see a longer time interval of the waveform. Decrease the scale to see it in more detail. The scale has a point that remains fixed on the screen when the scale value is changing - the reference point.

Remote command:
`TIMebase:SCALe` on page 1327

Acquisition time
Shows the time of one acquisition, that is the time across the 10 divisions of the diagram:

Acquisition time = Time scale * 10 divisions

Changing the acquisition time changes the time scale too.

Remote command:
`TIMebase:RANGe` on page 1328

Position
Defines the time distance between the reference point and the trigger point (the zero point of the diagram). If you want to see a section of the waveform some time before or after the trigger, enter this time as horizontal position. The requested waveform section is shown around the reference point. Use positive values to see waveform sections after the trigger - the waveform and the diagram origin move to the left.

See also "Reference point" on page 126.

Remote command:
`TIMebase:HORizontal:POSition` on page 1328

Reference point
Sets the position of the reference point in % of the screen. The reference point marks the rescaling center of the time scale. It is indicated by a grey triangle outline at the top of the diagram. If you modify the time scale, the reference point remains fixed on the screen, and the scale is stretched or compressed to both sides of the reference point.

Remote command:
`TIMebase:REFerence` on page 1328

Restrict horizontal position to acquisition range
If enabled, the horizontal position cannot be set outside the visible waveform diagram.

Remote command:
`TRIGger<m>:OFFSet:LIMited` on page 1329

Refence Clock
Opens a dialog for configuring the "Refence Clock" settings, see Chapter 5.3.2, "Reference clock (OCXO)", on page 130.

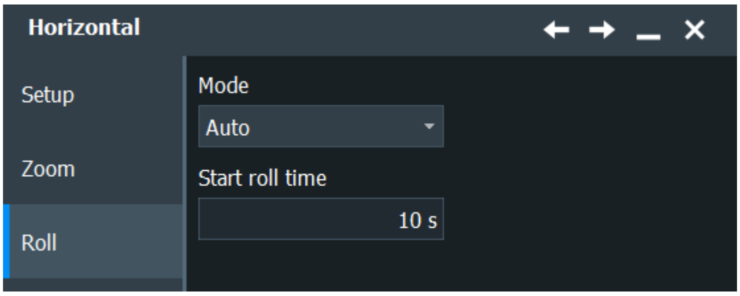
Skew
Opens a dialog for configuring the "Skew" settings, see Chapter 5.3.1, "Skew", on page 128.

5.2.2 Zoom settings

The zoom settings are described in Chapter 7.1, "Zoom", on page 241.

5.2.3 Roll settings

Access: "Menu" > "Horizontal" > "Roll" tab.



In this dialog, you can define the roll mode.

Roll mode
Selects, if the roll mode is set automatically by the instrument or if it is turned off.

In roll mode, the instrument shows the waveforms immediately, without waiting for the complete acquisition of the waveform record. If the time base is slow - at long time scale values - the roll mode saves waiting for the waveform display. The instrument displays newly acquired waveform points at the right edge of the display and moves the waveform to the left.

The roll mode has following restrictions:
- Roll mode disables persistence
- History is not available
- Event actions are not possible

The instrument activates the roll mode automatically if the following conditions are fulfilled:

- Acquisition time exceeds the defined "Minimum acquisition time"
- Waveform arithmetic is disabled ("Off")
- Only one waveform per channel is active
- All channel waveforms are set to the same decimation mode, and only to one of these values: "Sample", "Peak detect", or "High res"
- All mask tests are disabled
- Fast segmentation is disabled
- Event actions are disabled
- FFT is disabled
- All serial buses are disabled
- All digital channels are disabled (MSO option R&S RTO-B1)
- No CDR jitter data is acquired
- No zone trigger is active

The roll mode depends also on sample rate and record length. In roll mode, the sample rate limit is 20 MSample/s. At 50 s, the resulting record length limit is 1000 MSample.

If the acquisition time is >50 s, the record length limit is effective, and the maximum sample rate depends on the acquisition time:

Sample rate ≤ 1000 MSample / Acquisition time.

If the acquisition time is <50 s, the maximum sample rate in roll mode depends on the number of active channels:

Sample rate = 20 MSample/s / Number of active wfms.

The corresponding maximum record length is:

*Record length ≤ 20 MSample/s * Acquisition time / Number of active wfms*.

Thus, the roll mode switches off, or it does not activate automatically if:

- The record length exceeds the limit at acquisition times >50 s.
- The sample rate exceeds the limit.
- Too many waveforms are active.

Remote command:
`TIMebase:ROLL:ENABle` on page 1332
`TIMebase:ROLL:STATe?` on page 1332

Start roll time
The instrument can activate the roll mode automatically if the Acquisition time exceeds the value given here.

Remote command:
`TIMebase:ROLL:MTIMe` on page 1332

5.3 Horizontal accuracy

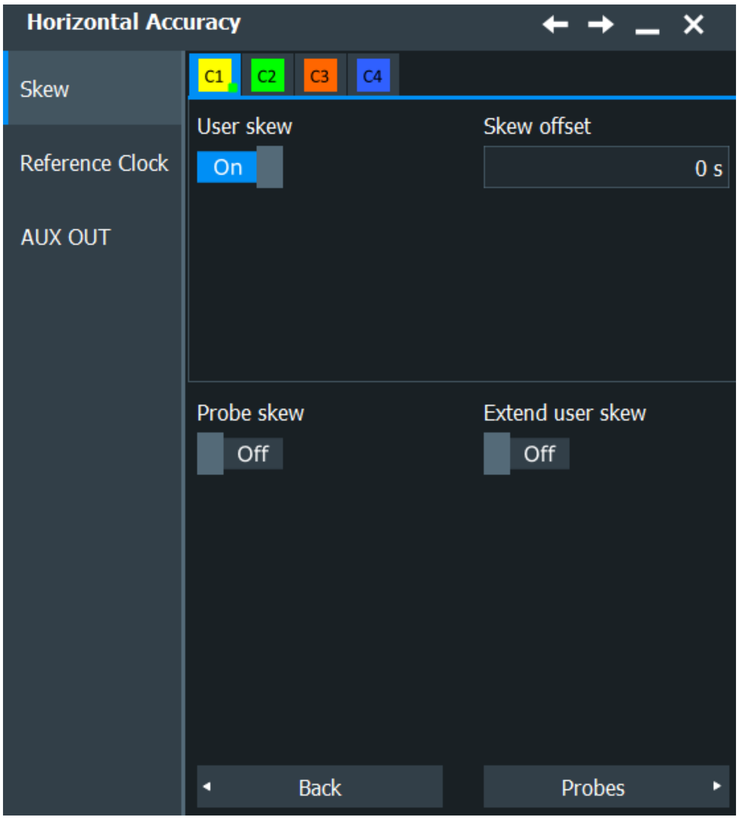
Access: "Menu" > "Horizontal" > "Setup" tab > "Skew".

The "Horizontal Accuracy" dialog box contains standard and optional settings to improve measurement and analysis accuracy and to reduce jitter effects.

5.3.1 Skew

Access: "Menu" > "Horizontal" > "Setup" tab > "Skew" > "Skew" tab.

Skew compensates signal propagation differences between channels caused by the different length of cables, probes, and other sources. Correct skew values are important for accurate triggering and timing relations between channels.



Make sure that the correct channel tab is selected.

User skew

If enabled, the "Skew offset" value is used for compensation.

Remote command:

`CHANnel<m>:SKEW:MANual` on page 1386

Skew offset

Sets a delay value, that is known from the circuit specifics but cannot be compensated by the instrument automatically. It affects only the selected input channel.

The offset range and possible values depend on Extend user skew.

Remote command:

`CHANnel<m>:SKEW:TIME` on page 1387

Probe skew

Measures the skew of all connected active probes and includes it in the total skew offset.

"Use probe group delay ..."

If enabled, the skew of all connected active probes is measured, displayed, and used for deskewing. The setting affects all active channels.

"Active probe"	Shows the type of the probe that is connected to the selected channel.
"Probe group delay"	Shows the result of the probe skew measurement on the selected channel.

Remote command:
`PROBe<m>:SKEState` on page 1387

Total skew offset
If Extend user skew is disabled, the sum of the measured "Probe group delay" and the "Skew offset" is shown. If "Use skew offset" is disabled, the skew offset is ignored.

If Extend user skew is enabled, only the probe skew is shown, and the skew offset is always ignored.

Extend user skew
Allows you to set higher value in "Skew offset" to compensate for the delay of the measurement setup. Without extension, the deskew range is ±100 ns, and delays shorter than the sample interval can be compensated.

With extension, the maximum delay is ±1 s, and the instrument can compensate complete samples. Compensation takes effect after the trigger. The maximum number of acquisitions in the memory is reduced.

Use the extended skew range to compare signals: One signal goes directly from the generator to the scope, and the other signal goes from the generator through the DUT to the scope. The delay of the DUT is higher than ±100 ns.

Remote command:
`CHANnel<m>:SKEW:EXTended` on page 1387

5.3.2 Reference clock (OCXO)

The OCXO reference clock requires option R&S RTO--B4.

Access: "Menu" > "Horizontal" > "Setup" tab > "Skew" > "Reference clock" tab.

The oven-controlled crystal oscillator (OCXO) produces a 10 MHz internal reference signal with precise and stable frequency. With OCXO, you can also use an external reference signal. The input and output connectors for the external reference signal are on the rear panel alongside the external trigger input.

Detected
Indicates if the OCXO is detected by the instrument.

Oven hot
Indicates when the oven has reached its nominal temperature and is operating with the specified accuracy.

External reference
Sets the frequency of an external reference input signal that is connected to the external reference input on the rear panel of R&S RTO. A frequency range from 1 MHz to 20 MHz is supported.

Remote command:
SENSe[:ROSCillator]:EXTernal:FREQuency on page 1390

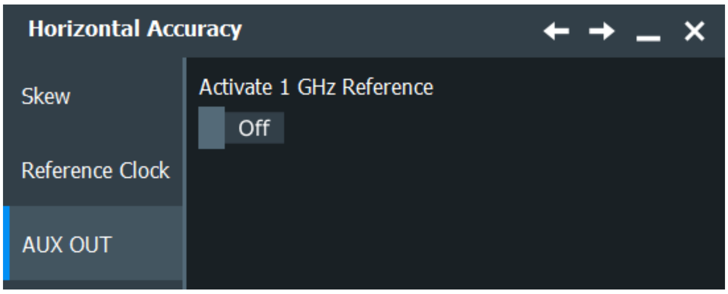
Use external reference
Enables the use of the external reference signal instead of the internal OCXO reference.

If an external reference is used, the frequency of the reference output signal is the same as of the reference input signal. Otherwise, the frequency of the reference output signal is 10 MHz, that is the frequency of the OCXO.

Remote command:
SENSe[:ROSCillator]:SOURce on page 1390

5.3.3 AUX OUT

Access: "Menu" > "Horizontal" > "Setup" tab > "Skew" > "AUX OUT" tab.



Activate 1GHz Reference
Enables the 1 GHz reference signal and sends it to the [Aux Out] connector. The signal is required for performance test to measure the frequency of the internal calibration signal.

Remote command:
CALibration:SOURce:FREQuency on page 1388
CALibration:SOURce:STATe on page 1388

5.4 Acquire settings

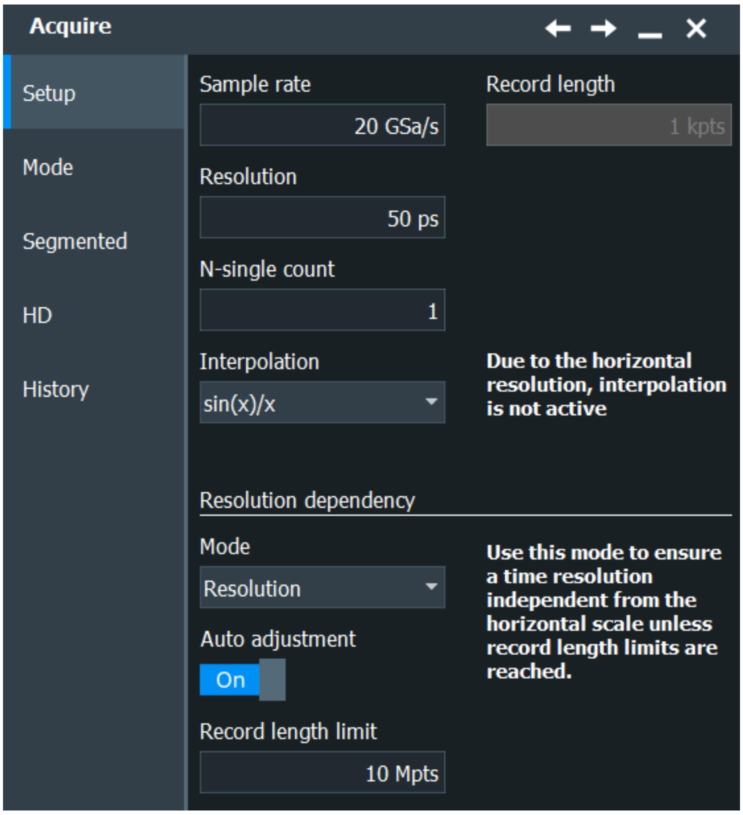
The "Acquire" dialog provides the acquisition configuration for channel and spectrum waveforms.

5.4.1 Setup settings

The "Setup" tab in the "Acquire" dialog provides the settings for the time axis.

For background information, see Chapter 5.1.3, "Horizontal system", on page 120.

Access: "Menu" > "Acquire" > "Setup" tab.



Sample rate

Sets the number of captured waveform points per second. It considers the samples of the ADC, and the reduction of waveform points by decimation.

If interpolation is not active, the sample rate is the reciprocal value of the resolution and thus also depends on the acquisition time and the record length.

If interpolation is active, the sample rate is limited to the ADC sample rate.

See also:
- Chapter 5.1.2, "Sampling and acquisition", on page 116
- Chapter 5.1.3, "Horizontal system", on page 120

Remote command:
`ACQuire:SRReal` on page 1331

Record length

Indicates the number of waveform samples that build the waveform across the acquisition time.

Remote command:
`ACQuire:POINts[:VALue]` on page 1331

Resolution

Sets the time between two waveform samples. A fine resolution with low values produces a more precise waveform record.

Remote command:
`ACQuire:RESolution` on page 1331

Average count (N-single count)

Access:
- "Menu" > "Acquire" > "Setup" tab > "Average count (N-single count)"
- "Menu" > "Acquire" > "Segmented" tab > disable "Acquire maximum" > "Required"
- [Math] > "Setup" tab > "Mode" is not "Off" > "Average count"

The acquisition and average count has several effects:
- It sets the number of waveforms acquired with [Single]
- It defines the number of waveforms used to calculate the average waveform.
  Thus, the instrument acquires sufficient waveforms to calculate the correct average if "Average" is enabled for waveform arithmetic. The higher the value is, the better the noise is reduced.
- It sets the number of acquisitions to be acquired in a fast segmentation acquisition series. Thus, you can acquire exactly one fast segmentation acquisition series with [Single].
  If fast segmentation is enabled and configured to acquire the maximum number of acquisitions, the acquisition count is set to that maximum number and cannot be changed.
  See also "Number of acquisitions" on page 138.
- It is the "Finished" criteria for the state of a mask test.

Remote command:
`ACQuire:COUNt` on page 1336

Interpolation

Selects the interpolation method. If the defined "Sample rate" is higher than the ADC sample rate, interpolation adds points between the captured samples of the waveform by various mathematic methods.

"Linear"	Two adjacent ADC sample points are connected by a straight line, the interpolated points are located on the line. You see a polygonal waveform similar to the real signal, and also the ADC sample points as vertexes.
"sin (x)/x"	Two adjacent ADC sample points are connected by a sin(x)/x curve, and also the adjoining sample points are considered by this curve. The interpolated points are located on the resulting curve. This interpolation method is precise and shows the best signal curve.
"Sample/Hold"	The ADC sample points are displayed like a histogram. For each sample interval, the voltage is taken from the sample point and considered as constant, and the intervals are connected with vertical lines. Thus, you see the discrete values of the ADC - the measured samples.

Remote command:
`ACQuire:INTerpolate` on page 1333

Resolution dependency

Selects the resolution dependency settings

Mode ← Resolution dependency

You can choose to keep constant either the resolution or the record length when you adjust the time scale or acquisition time.

"Resolution"	With constant resolution, increasing the time scale also increases the record length, and vice versa. You can limit the record length to a maximum value.
"Record length"	With constant record length, increasing the time scale coarsens the resolution, that is, the time between two waveform samples gets longer.

Remote command:
`ACQuire:POINts:AUTO` on page 1329

Auto adjustment ← Resolution dependency
Prevents undersampling and ensures a sufficient resolution to acquire the correct waveform if the time scale is changed. The setting takes effect if the changed parameter - resolution or record length - reaches a limit. The instrument automatically keeps this parameter constant at its limit, and changes the other parameter regardless of the "Resolution / Record length" setting.

See also: Record length

Remote command:
`ACQuire:POINts:AADJust` on page 1330

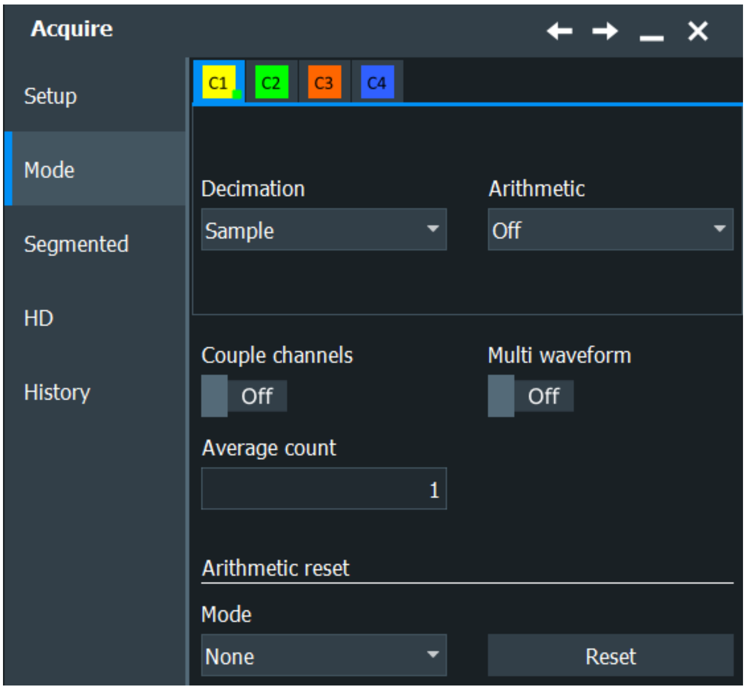
Record length ← Resolution dependency
Sets a limit for the record length to prevent very large records. This value is only available if "Auto adjustment" is on and a constant resolution is selected. If you increase the time scale, the resolution remains constant and the record length increases until the limit is reached. Further increase of the time scale changes the resolution and keeps the record length limit.

See also:
- Record length
- Auto adjustment

Remote command:
`ACQuire:POINts:MAXimum` on page 1330

5.4.2 Mode settings

Access: "Menu" > "Acquire" > "Mode" tab.



Mode
Selects the decimation mode. Decimation reduces the data stream of the ADC to a stream of waveform points with lower sample rate and a less precise time resolution. The R&S RTO uses decimation, if the waveform "Sample rate" is less than the ADC sample rate. In this case, interpolation is not possible.

The decimation mode is waveform-specific, you can select another mode for each waveform.

There are different methods to define the recorded waveform point out of n sample points:

"Sample"	One of n samples in a sample interval of the ADC is recorded as waveform point, the other samples are discarded. The time between the two adjacent waveform points is exactly the resolution. Very short glitches might remain undiscovered by this method.
"Peak detect"	The minimum and the maximum of n samples in a sample interval are recorded as waveform points, the other samples are discarded.
"High res"	The average of n sample points is recorded as one waveform sample. Averaging reduces the noise, the result is a more precise waveform with higher vertical resolution.
"RMS"	The waveform point is the root mean square of n sample values. Thus, the RMS value reflects the instantaneous power. This arithmetic mode is used to average a measured power waveform. Linear averaging of power signals causes an error dependent on the noise of the signal to be averaged.

Remote command:
`CHANnel<m>[:WAVeform<n>]:TYPE` on page 1334

Wfm Arithmetic
Waveform arithmetic builds the resulting waveform from several consecutive acquisitions of the signal. The arithmetic works with interpolated and decimated waveforms.

This setting is waveform-specific.

The methods are:

"Off"
The data of only one acquisition is recorded according to the decimation settings. In effect, no waveform arithmetic is processed.

"Envelope"
Detects the minimum and maximum values in a sample interval over several acquisitions. Each acquisition is done in the "Peak detect" decimation mode, and the most extreme values for all acquisitions build the envelope. The resulting diagram shows two envelope waveforms: the minimums (floor) and maximums (roof).
The envelope is built until the restart criterion is reached, see "Arithmetic Reset > Mode" on page 137.

Note: If you change from "Envelope" to "Off", make sure to set also the "Mode" to the required value.

"Average"
The average is calculated from the data of the current acquisition and several acquisitions before. The method reduces random noise and other heterodyne signals. It requires a stable, triggered and periodic signal for correct function.
The number of acquisitions for average calculation is defined with "Average count"
The "Auto reset mode" defines the restart condition.

Remote command:
`CHANnel<m>[:WAVeform<n>]:ARIThmetics` on page 1335

Couple channels
Sets the acquisition mode and the waveform arithmetic of all channels to the last set value.

If the acquisition settings are coupled, "Multi waveform" is not available, only one waveform per channel can be used.

Remote command:
`ACQuire:CDTA` on page 1333

Multi waveform
For each channel, up to three waveforms can be shown and analyzed. The decimation mode and the waveform arithmetic are specific for each waveform. So you can analyze several aspects of the signal: For example, waveform1 shows the peaks, and waveform2 shows the average of the signal.

Remote command:
`ACQuire:MUWaveform` on page 1334

Average count (N-single count)
Access:
- "Menu" > "Acquire" > "Setup" tab > "Average count (N-single count)"
- "Menu" > "Acquire" > "Segmented" tab > disable "Acquire maximum" > "Required"
- [Math] > "Setup" tab > "Mode" is not "Off" > "Average count"

The acquisition and average count has several effects:

- It sets the number of waveforms acquired with [Single]
- It defines the number of waveforms used to calculate the average waveform.
  Thus, the instrument acquires sufficient waveforms to calculate the correct average if "Average" is enabled for waveform arithmetic. The higher the value is, the better the noise is reduced.
- It sets the number of acquisitions to be acquired in a fast segmentation acquisition series. Thus, you can acquire exactly one fast segmentation acquisition series with [Single].
  If fast segmentation is enabled and configured to acquire the maximum number of acquisitions, the acquisition count is set to that maximum number and cannot be changed.
  See also "Number of acquisitions" on page 138.
- It is the "Finished" criteria for the state of a mask test.

Remote command:
`ACQuire:COUNt` on page 1336

Arithmetic Reset > Mode

Defines when the envelope and average evaluation restarts.

"None"	No restart, the number of acquisitions considered by the waveform arithmetics is not limited.
"Time"	Restarts the envelope and average calculation after the time defined in "Time".
"Waveforms"	Restarts the envelope and average calculation after a number of acquired waveforms defined in "Count".

Remote command:
`ACQuire:ARESet:MODE` on page 1336
`ACQuire:ARESet:TIME` on page 1336
`ACQuire:ARESet:COUNt` on page 1337

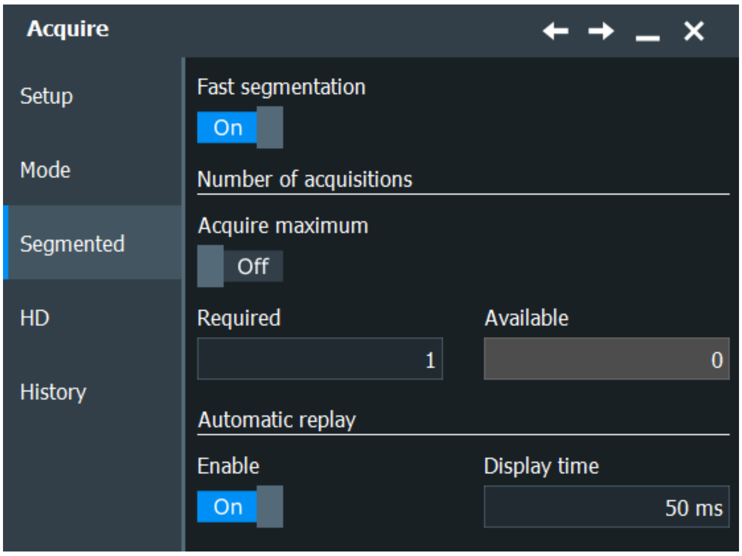
Reset now

Forces the immediate restart of the envelope and average calculation for all waveforms.

Remote command:
`ACQuire:ARESet:IMMediate` on page 1336

5.4.3 Segmented settings

Access: "Menu" > "Acquire" > "Segmented" tab.



Fast segmentation
Switches the fast segmentation mode on and off.

Remote command:
`ACQuire:SEGMented:STATe` on page 1337

Number of acquisitions
You can define the number of acquisitions to be stored in a fast segmentation acquisition series:

- Acquire the maximum number of acquisitions that can be stored in the sample memory.
  To acquire the maximum number, enable "Acquire maximum". The maximum number of acquisitions is shown in the "Required" field.
- Acquire a given number of acquisitions.
  Enter the number in the "Required" field.

The acquisition count (Average count (N-single count)) is always set to the required number of acquisitions. Thus you can acquire exactly one fast segmentation acquisition series with [Single]. The [Run Stop] key works in the same way as [Single], it stops acquisition when the series is completed.

You can stop the running acquisition before the series is completed.

The number of acquired waveforms is shown in "Available" and can be displayed with "Show history".

Remote command:
`ACQuire:SEGMented:MAX` on page 1337

Enable automatic replay
If enabled, the instrument starts processing and displaying the data when the acquisition series is captured completely. Depending on the number of acquisitions, it can take some time until the acquisition series is displayed. If the setting is disabled, the instrument only captures the data and stores it in the sample memory.

Remote command:
`ACQuire:SEGMented:AUToreplay` on page 1338

Display time
Defines the display speed of the fast segmentation acquisition series. Display starts after the series has been captured completely.

5.4.4 High definition mode

The high definition mode offers up to 16 bits of vertical resolution. Higher vertical resolution reduces quantization noise and acquires waveforms of higher accuracy with finer details of the signal to be seen.

The number of vertical resolution bits defines the number of vertical levels that the acquisition samples are mapped to (quantization). 16 bits of resolution represent 65536 voltage quantization levels, while 8 bits of resolution represent only 256 voltage levels. The waveform values are recorded with 16 bit word length, except for peak detect decimation.

The higher vertical resolution is achieved by applying a digital low pass filter (DSP filter) to the output of the ADC, which reduces the bandwidth of the signal. Increasing the bandwidth reduces the resulting digital resolution. The high definition is also applied to the digital trigger, thus the R&S RTO can trigger with the same high resolution with which they can display signals.

High definition can be used, for example, to measure slow pulses with high accuracy, or to analyze AM signals with very low modilation index, as used in radar.

See also:
- Chapter 5.1.1, "Vertical system", on page 114
- Chapter 5.1.2, "Sampling and acquisition", on page 116

5.4.4.1 High definition settings

Access: "App Cockpit" menu > "HD"

High definition is a special acquisition mode of the oscilloscope. This mode has only one setting - the filter bandwidth.

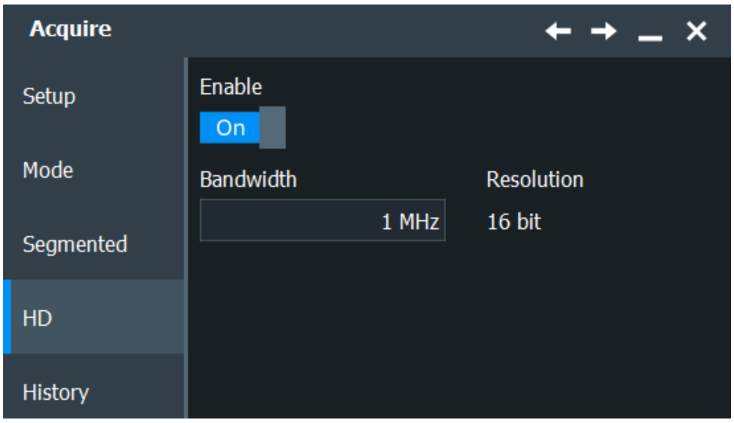


Figure 5-5: Setting the instrument into high definition mode

Enable

Enables the higher digital resolution, up to 16 bit.

Remote command:
HDEFinition:STATe on page 1389

Bandwidth

Sets the filter bandwidth for the high definition mode.

The maximum filter bandwidth depends on the instrument bandwidth.

Instrument bandwidth	Maximum filter bandwidth
600 MHz	500 MHz
≥ 1 GHz	1 GHz

Remote command:
HDEFinition:BWIDth on page 1389

Resolution

Shows the resulting vertical resolution in high definition mode. The higher the filter bandwidth, the lower the resolution. For details, refer to the R&S RTO Specifications.

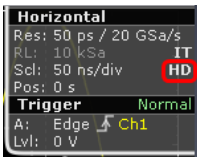
Remote command:
HDEFinition:RESolution? on page 1389

5.4.4.2 Effects of the high definition mode

The high definition mode has several effects:

Acquisition

The active high definition mode is indicated by "HD" in the "Acquisition" label.



The high definition mode works with half the realtime sample rate. For FFT, the instrument halves this sample rate again.

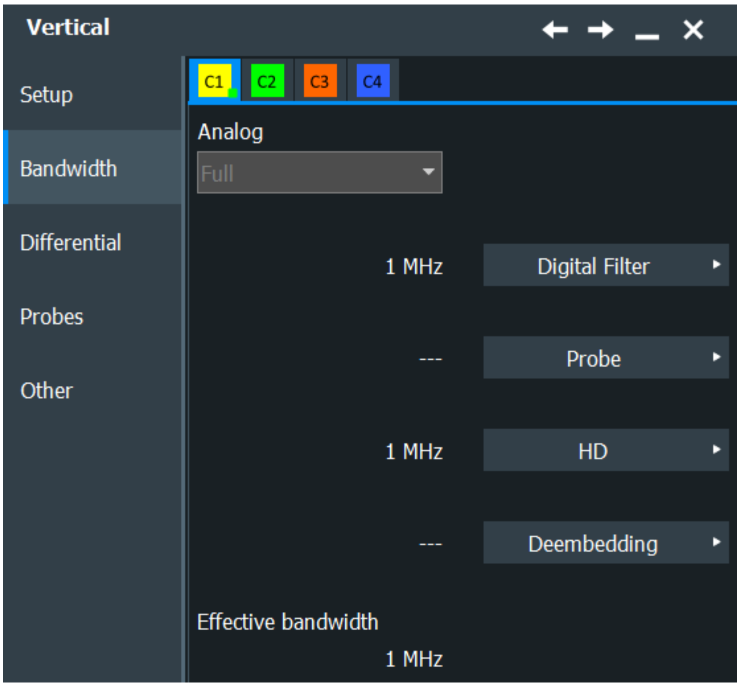
The waveform values are recorded with 16-bit word length, except for peak detect decimation (2 values with 8 bit).

Vertical system

The current bandwidth is shown in the channel label.

In the "Vertical" > "Bandwidth" dialog (C<x>), the "Bandwidth" setting is not available because the bandwidth is set by the high definition filter.

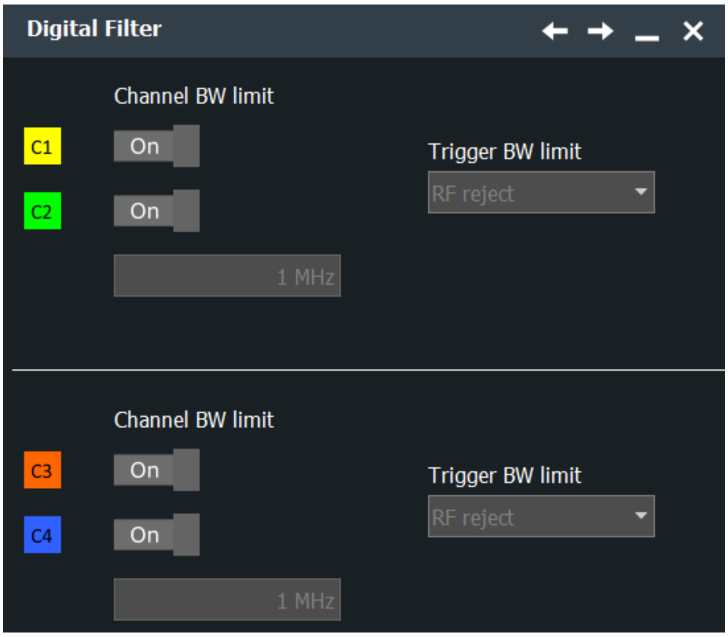
The minimum vertical scale is 500 µV/div instead of 1 mV/div in normal mode.



Digital filter

The digital filter settings are set automatically.

You can change the high definition "Bandwidth" in the "Acquire" > "HD" dialog.



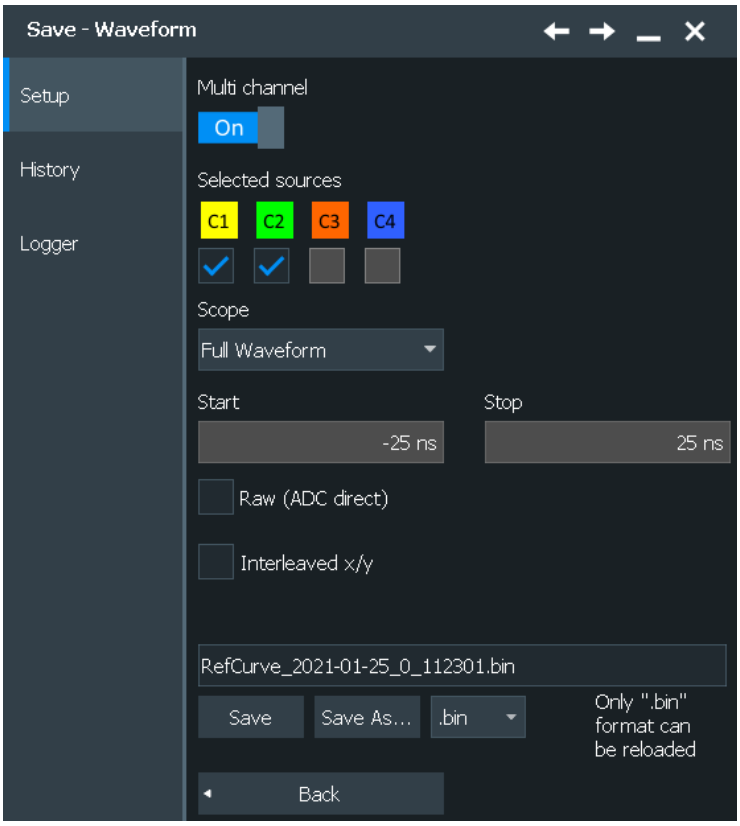
History

Due to the 16-bit word length, the history depth is reduced, less waveforms are saved than in normal mode.

Export

In high definition mode, waveform data in raw format is exported to file with 16-bit word length, except for peak detect decimation (2 values with 8 bit). In addition, you can define the byte order of the data words.

You can define additional export settings in the "Menu" key >"Save/Recall" key > "Save" tab > "Waveform" > "Setup" tab.



See:
- "Raw (ADC direct)" on page 458
- "Interleaved x/y" on page 459
- "Byte order" on page 1242

If you use remote control commands to transfer data to a controlling computer, set the data format to `INT,16` to transfer the complete data words (see `FORMat[:DATA]` on page 1293).

Further restrictions

The IQ mode (option R&S RTO--K11) is not available if high definition mode is active.

5.4.5 History settings

The history settings are described in Chapter 7.4.2, "History setup", on page 279.

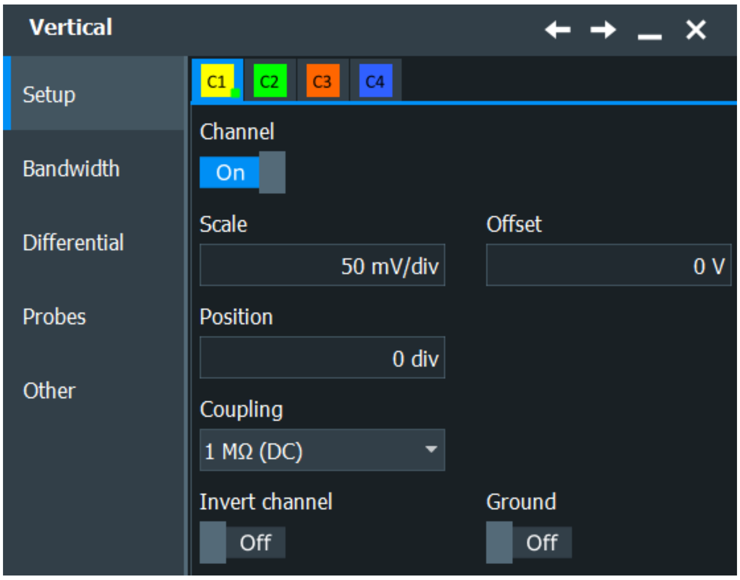
5.5 Vertical setup

The "Vertical" dialog box contains all channel-dependent settings and information.

5.5.1 Setup settings

Access: "Menu" > "Vertical" > "Setup"

The "Setup" tab provides all basic vertical settings. The channels are listed in horizontal subtabs. Make sure to select the correct channel tab before you enter the settings.



Channel On/Off

Switches the channel signal on or off. The signal icon appears on the signal bar. The waveform of the last acquisition is displayed in the diagram.

Remote command:
`CHANnel<m>:STATe` on page 1338

Scale

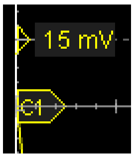
Defines the vertical scale in Volts per division. Increasing the scale compresses the display of the signal.

Remote command:
`CHANnel<m>:SCALe` on page 1339

Offset

The offset voltage is subtracted to correct a signal with DC offset. The vertical center of the selected channel is shifted by the offset value and the signal is repositioned within the diagram area. Negative offset values move up the waveform, positive values move it down.

The offset of a signal is determined and set by the autoset procedure. The current value is shown in the waveform label, and it is marked by a small triangle in the grid.



If a Rohde & Schwarz differential probe is connected, the offset is the differential offset.

If a Rohde & Schwarz modular probe is connected, the offset of the selected probe mode is used. For example, in CM mode, the offset is the common mode offset.

By default, the horizontal grid axis remains in the center when the offset is changed. To shift the axis together with the waveform, disable Keep Y-grid fixed in ""Settings" > Appearance > Grid".

Remote command:
`CHANnel<m>:OFFSet` on page 1340

Position

Moves the selected signal up or down in the diagram. The visual effect is the same as for Offset but the waveform is adjusted later in the signal flow. While the offset sets a voltage, position is a graphical setting given in divisions.

By default, the horizontal grid axis remains in the center when the offset is changed. To shift the axis together with the waveform, disable Keep Y-grid fixed in ""Settings" > Appearance > Grid".

Remote command:
`CHANnel<m>:POSition` on page 1340

Coupling

Selects the connection of the channel signal. The coupling determines what part of the signal is used for waveform analysis and triggering.

In addition to coupling, the signal can be filtered for high frequency rejection, see Chapter 5.5.5, "Digital filter setup", on page 148.

"DC 50 Ω"	Connection with 50 Ω termination, passes both DC and AC components of the signal.
"DC 1 MΩ"	Connection with 1 MΩ termination, passes both DC and AC components of the signal.
"AC"	Connection with 1 MΩ termination through DC capacitor, removes DC and very low-frequency components. If AC coupling is set, the attenuation of passive probes has no effect, and voltage is applied to the instrument with factor 1:1. Observe the voltage limits, otherwise you can damage the instrument.

Remote command:
`CHANnel<m>:COUPling` on page 1338

Invert channel
Turns the inversion of the signal amplitude on or off. To invert means to reflect the voltage values of all signal components against the ground level. If the inverted channel is the trigger source, the instrument triggers on the inverted signal.

You can use inversion, for example, to switch the polarity of a differential signal without changing the probe connections.

Remote command:
`CHANnel<m>:INVert` on page 1341

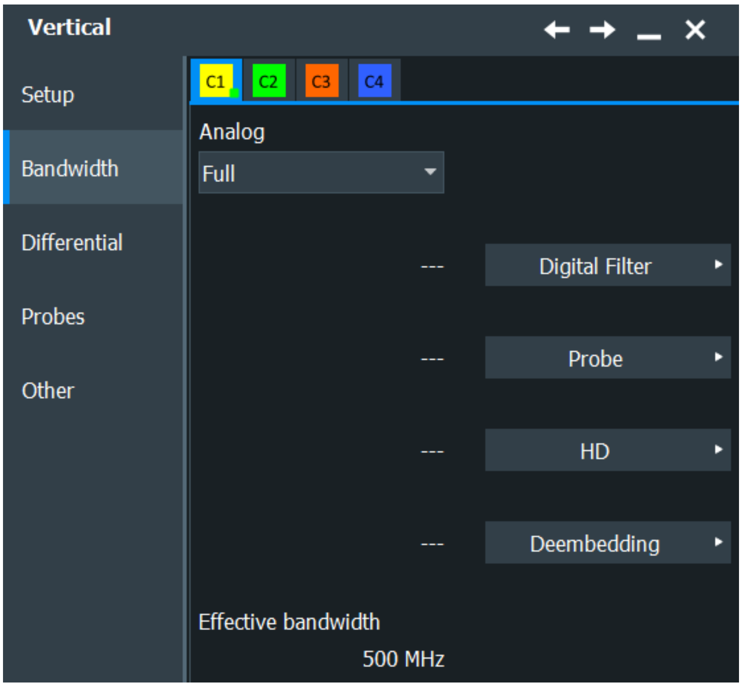
Ground
Connects the input to the ground.

Remote command:
`CHANnel<m>:GND` on page 1339

5.5.2 Bandwidth settings

Access: "Menu" > "Vertical" > "Bandwidth"

The "Bandwidth" tab provides all settings that affect the bandwidth of the measurement system. The channels are listed in horizontal subtabs. Make sure to select the correct channel tab before you enter the settings.



Analog
Selects the bandwidth limit.

The specified bandwidth indicates the range of frequencies that the instrument can acquire and display accurately with less than 3dB attenuation. The probe has also a limited bandwidth and thus affects the resulting system bandwidth.

The current bandwidth is shown on the signal icon. At full bandwidth, the displayed bandwidth can be less than the instrument bandwidth depending on the number of active channels and other settings.

See also: Chapter 5.1.1.3, "Bandwidth", on page 116

"Full" At full bandwidth, all frequencies in the specified range are acquired and displayed. Full bandwidth is used for most applications.

"20 MHz, 200 MHz, 800 MHz"
Frequencies above the selected limit are removed to reduce noise at different levels.
The "800 MHz" filter is available for 50 Ω coupling on scopes with ≥ 1 GHz instrument bandwidth.

Remote command:
`CHANnel<m>:BANDwidth` on page 1341

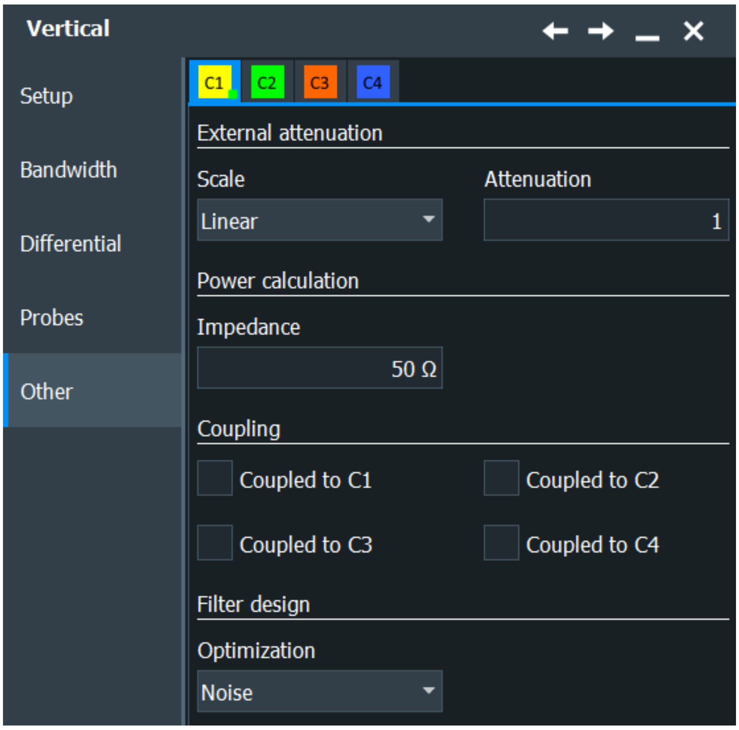
Effective bandwidth
Displays the effective bandwidth.

5.5.3 Probes settings

See Chapter 5.6, "Probes", on page 150.

5.5.4 Other settings

Access: "Menu" > "Vertical" > "Other"



External attenuation: Scale, Attenuation
Consider a voltage divider that is part of the DUT before the measuring point. The external attenuation is included in the measurement, and the instrument shows the results that would be measured before the divider. External attenuation can be used with all probes.

"Scale" Select linear or logarithmic attenuation scale.

"Attenuation" Enter the attenuation of the voltage divider according to the selected scale. The conversion from linear to logarithmic values depends on the "Vertical unit" of the probe:
For voltage-based unit (V and A):
*attenuation (dB) = 20 * log_{10}(attenuation factor)*
For power-based unit (W):
*attenuation (dB) = 10 * log_{10}(attenuation factor)*

Remote command:
`CHANnel<m>:EATScale` on page 1348
`CHANnel<m>:EATTenuation` on page 1348

Impedance
Sets the impedance of the channel for power calculations and measurements.

Remote command:
`CHANnel<m>:IMPedance` on page 1342

Coupled to C1 / C2 / C3 / C4
Channel coupling sets the vertical settings of the coupled channels to the values of the active channel. If you want to have the same vertical settings for two or more channels, you can set them at once by coupling these channels.

Channel coupling affects all vertical settings that are adjusted in the "Settings" tab: vertical scale, position, offset, coupling, and ground.

Remote command:
`CHANnel<m>:CPLing` on page 1342

5.5.5 Digital filter setup

Access: "Menu" > "Vertical" > "Bandwidth" tab > "Digital Filter"

After processing by the A/D converter, the channel and trigger signals are digitized signals. These digitized signals can be filtered to reject high frequency - also known as Digital Signal Processing (DSP). You can filter the acquisition channels and the trigger channel signal.

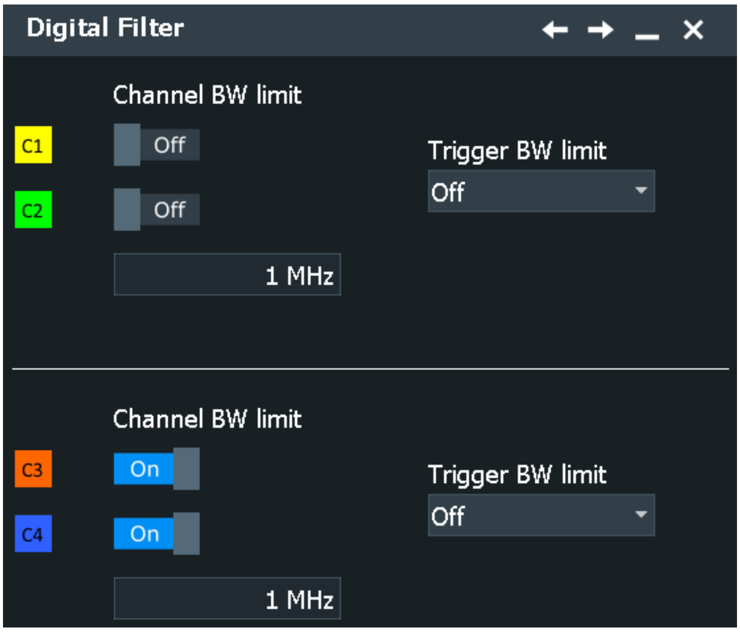
One filter is applied to a pair of channels. If you filter only on the input channels, you can apply different filters to the channels - one filter for channels 1 and 2 and another filter for channels 3 and 4.

If you filter the trigger channel, the same filter must be used for the input channels to ensure that all signals suit for analysis. The instrument offers only permitted combinations and triggers on the filtered signal.

In high definition mode, digital filter settings are enabled automatically. You can change the high definition bandwidth in the "Digital Filter" Setup, which is applied to the channels.

Example:

"RF reject" for the trigger signal ensures that triggering is not caused by unexpected glitches.



Filter on channel: On | Off

Enables the DSP filter for the correspondig input channel.

Remote command:
CHANnel<m>:DIGFilter:STATe on page 1385

Trigger BW limit

Selects the filter for the trigger channel. Other channels must use the same filter, or proceed unfiltered.

"Off" The trigger signal is not filtered, and the acquisition channels can be filtered independently.

"RF reject" Frequencies higher the "Channel BW limit" are rejected, lower frequencies pass the filter.

Remote command:
TRIGger<m>:COUPling on page 1385

Channel BW limit

If "Trigger BW limit" is "Off", it sets the limit frequency of the lowpass filter for input channels. One filter is applied to a pair of channels.

If "Trigger BW limit" is "RF reject", it sets the limit frequency that is applied to the trigger channel and to the input channels enabled for filtering.

Remote command:
`CHANNel<m>:DIGFilter:CUToff` on page 1385
`TRIGger<m>:RFReject` on page 1386

5.6 Probes

With R&S RTO digital oscilloscopes, you can use various probe types. Mostly these probes are passive and active voltage probes. The "Probes" dialog provides all probe-relevant information.

The instrument can detect many probes and read out the probe-specific parameters, for example, bandwidth and attenuation.

In the "Probes" tab, you find all settings that are relevant for the connected probe.

Access: "Menu" > "Vertical" > "Probes"

The functionality on the "Probes" tab changes according to the type of the attached probe. Probes with Rohde & Schwarz probe interface (probe box), and also many other passive voltage probes, are recognized by the instrument. The R&S RTO reads out the main characteristics of the probe and displays them. Other probes cannot be detected, but their characteristics are known to the instrument. These known probes are called "Predefined probes". Probes that are not recognized automatically and not predefined are unknown probes, they require manual setting of measurement unit and attenuation.

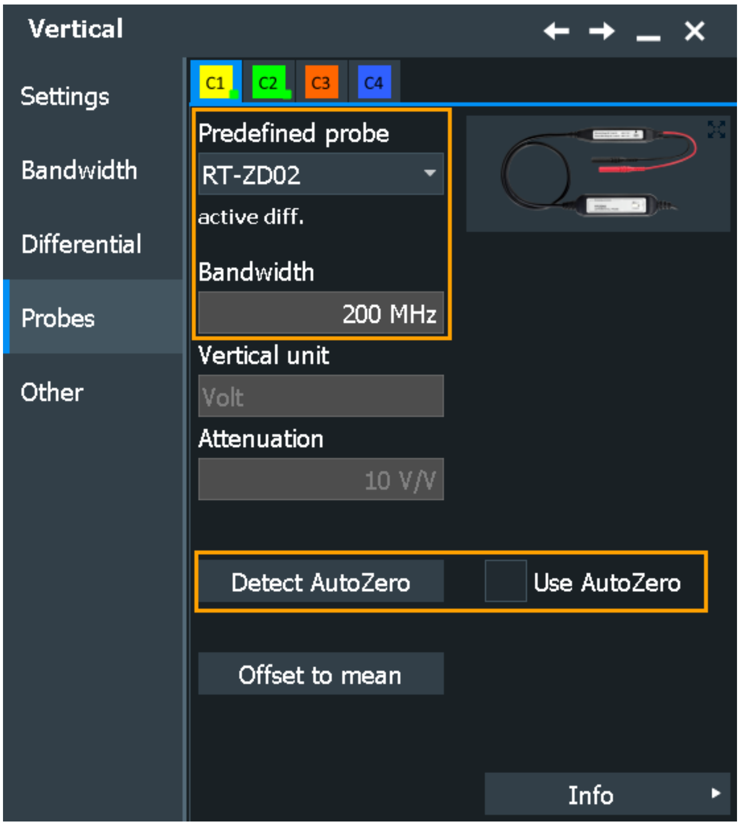
Before you adjust the settings, select the correct channel tab.

Additional information is given in the "Probe Attributes" and "Calibration Results" tabs. For background information, see Chapter 5.1.4, "Probes", on page 121.

This chapter has the following sections:

5.6.1 Shared probe settings

Some of the settings in the "Probes" tab are available for all probes.



An external attenuation can be set on the "Other" tab, see "External attenuation: Scale, Attenuation" on page 148.

The shared probe settings are:

Type, Name, Bandwidth

The fields show the characteristics of a recognized or predefined probe for information. If the instrument cannot recognize the probe, and the probe is not known, the "Type" is "None", and the other fields are empty.

Remote command:
`PROBe<m>:SETup:TYPE?` on page 1346
`PROBe<m>:SETup:NAME?` on page 1346
`PROBe<m>:SETup:BANDwidth?` on page 1347

Detect AutoZero, Use AutoZero

Differences in DUT and oscilloscope ground levels can cause larger zero errors, which affect the waveform. If the DUT is ground-referenced, the AutoZero function corrects the zero error of the probe to optimize measurement results at small signal levels. The validation limit depends on the probe attenuation because probes with high attenuation often have to compensate high offsets. AutoZero detects offset values even when the signal is out of the current measurement range.

To correct the zero error of voltage probes, short the signal pin and the ground pin together and connect them to the ground of the DUT. Then tap "Detect AutoZero". While the alignment is running, the instrument switches to DC coupling to display the waveform correctly.

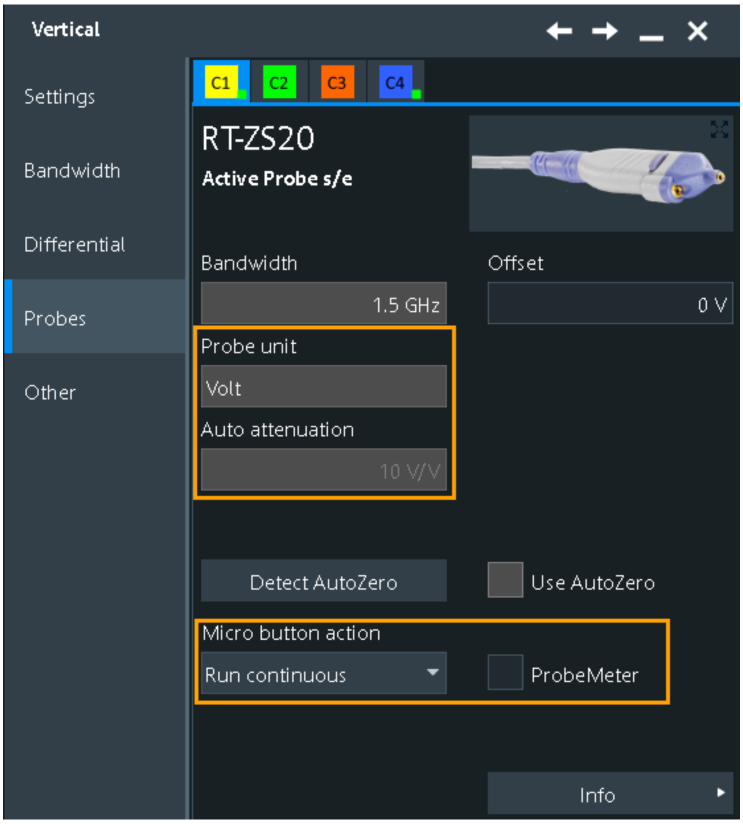
To include the measured offset in measurement results, enable "Use AutoZero".

If a current probe is connected, the function demagnetizes the probe's sensor head and sets the waveform to zero position. See "Detect AutoZero" on page 172.

Remote command:
`PROBe<m>:SETup:OFFSet:AZERo` on page 1347
`PROBe<m>:SETup:OFFSet:USEautozero` on page 1347

5.6.2 Settings for the R&S probe interface (voltage probes)

Active voltage probes with Rohde & Schwarz probe interface provide special features: the micro button and the ProbeMeter. Furthermore, the R&S RTO can read out the attenuation of the probe.

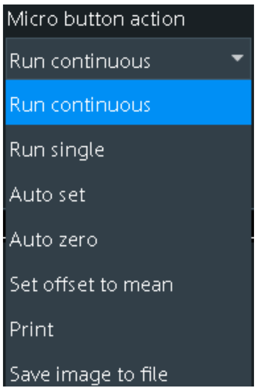


The settings for active voltage probes with Rohde & Schwarz probe interface are:

Probe unit, Auto attenuation
If the probe is recognized by the R&S RTO, the instrument reads the attenuation unit and value from the probe an displays them.

Remote command:
PROBe<m>:SETup:ATTenuation[:AUTO]? on page 1347

Micro button action
Active voltage probes with Rohde & Schwarz probe interface have a configurable micro button on the probe head. Pressing this button, you start an action on the instrument directly from the probe. The button is disabled during internal automatic processes, for example, during self-alignment, autoset, and find level.



Select the action that you want to start from the probe:

"Run Continuous"
Is the default assignment. Starts or stops the acquisition (same as Run Stop key).

"Run single" Starts a defined number of acquisitions (same as [Single] key).

"Auto set" Starts the autoset procedure (same as Autoset key).

"AutoZero" Starts an auto zero measurement, see "Detect AutoZero, Use AutoZero" on page 151.

"Set offset to mean"
Performs an automatic compensation for a DC component of the input signal using the result of a background mean measurement.
See: "Offset to mean" on page 169.

"Save image to file"
Saves the current display as image according to the image settings.
See also Chapter 12.3, "Screenshots", on page 471.

"No action" Select this option to prevent unwanted actions due to unintended usage of the micro button.

"Find trigger level" Sets the trigger level automatically to *0.5 * (MaxPeak – MinPeak)*.

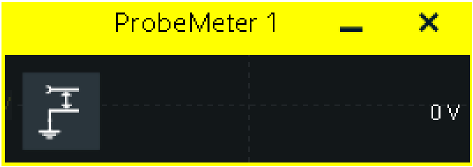
"Create report" Creates and saves a report using the report settings. See also Chapter 12.4.1, "Report settings", on page 476.

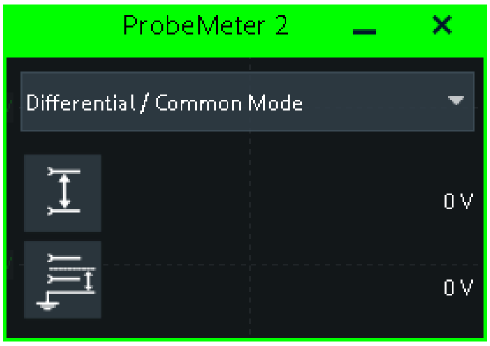
"Probe mode" Only available if a R&S RT-ZM modular probe is connected. Sets the measurement mode of the modular probe. See also "Probe mode" on page 163.

"Probe Setup" Opens the "Probes Setup" dialog box.

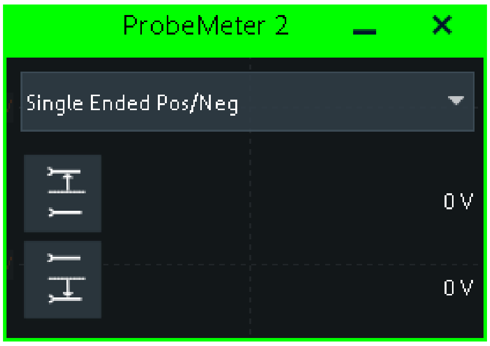
Remote command:
`PROBe<m>:SETup:MODE` on page 1349

ProbeMeter
The integrated R&S ProbeMeter of active voltage probes with Rohde & Schwarz probe interface is a voltmeter. It measures DC voltages between the probe tip and ground connection or between the probe tips with very high precision. The R&S ProbeMeter enables ground-referenced measurements of voltages. The measurement is performed continuously and in parallel to the measurements of the oscilloscope.

- **"Probemeter"**
  Select "Probemeter" to activate the integrated R&S ProbeMeter of active R&S probes. The measured voltages are displayed in the "ProbeMeter" result box on the screen.
- **ProbeMeter measurement results of single-ended active R&S probes**
  Measures the voltage between the probe tip and the ground.
  

- **ProbeMeter measurement results of differential and modular R&S probes**
  You can select the voltage to be measured by the differential active probe:
  – "Differential / Common Mode":
    Differential voltage is the voltage between the positive and negative signal sockets.
    Common mode voltage is the mean voltage between the signal sockets and the ground socket. It measures the voltage level relative to ground, for example, to check the operating voltage window.
    

  – "Single Ended Pos/Neg": Measures the voltage between the positive/negative signal socket and the ground.



The ProbeMeter always measures the common mode and differential voltages. Single-ended voltages are calculated values:
$V_p = V_{cm} + 0.5 * V_{in}$ and $V_n = V_{cm} - 0.5 * V_{in}$

Remote command:
`PROBe<m>:PMETer:VISibility` on page 1350
`PROBe<m>:SETup:DISPlaydiff` on page 1350
`PROBe<m>:PMETer:RESults:SINGle?` on page 1350
`PROBe<m>:PMETer:RESults:POSitive?` on page 1352
`PROBe<m>:PMETer:RESults:NEGative?` on page 1351
`PROBe<m>:PMETer:RESults:DIFFerential?` on page 1351
`PROBe<m>:PMETer:RESults:COMMon?` on page 1351

5.6.3 Setup for passive probes

Passive probes are the most widely used probes for oscilloscope measurements. Passive probes require compensation.

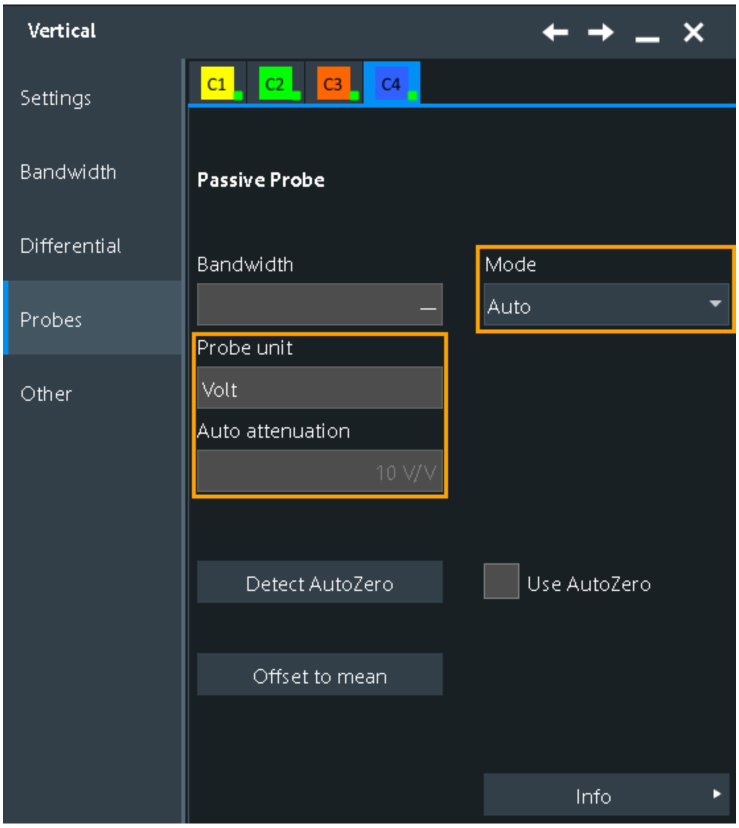


Figure 5-6: Probe setup for passive probe

The following shared probe settings are available:

- "Type, Name, Bandwidth" on page 151
- "Detect AutoZero, Use AutoZero" on page 151

If a passive probe is connected, the probe attenuation is read out and shown in the "Setup" tab:

- "Probe unit, Auto attenuation" on page 153

If you need to change the unit or attenuation, change the "Mode" to "Manual" and enter the correct values.

Mode

Defines how the attenuation of a passive probe is set.

"Auto" The instrument uses the values that are read out from the probe.

"Manual" You can define the attenuation unit and value.
See: "Vertical unit, Attenuation, Gain" on page 157

Remote command:
`PROBe<m>:SETup:ATTenuation:MODE` on page 1352

Vertical unit, Attenuation, Gain
If a predefined probe is connected and selected, the attenuation or gain values are shown.

For unknown probes and passive probes in manual mode, you can set user-defined values for unit, gain and attenuation.

If AC coupling is set, the attenuation of passive probes has no effect, and voltage is applied to the instrument with factor 1:1. Observe the voltage limits, otherwise you can damage the instrument.

Remote command:
`PROBe<m>:SETup:ATTenuation:UNIT` on page 1353
`PROBe<m>:SETup:ATTenuation:MANual` on page 1353
`PROBe<m>:SETup:GAIN:MANual` on page 1353

5.6.4 Setup for active voltage probes

Active voltage probes with Rohde & Schwarz probe interface have an integrated data memory that contains identification data and individual probe correction parameters. The R&S RTO can detect these probes and read out the data. Furthermore, these probes have a micro button and a ProbeMeter.

Active voltage probes that are offered by Rohde & Schwarz but not equipped with a Rohde & Schwarz probe interface are known to the R&S RTO as predefined probes, see Chapter 5.6.6, "Setup for predefined probes", on page 167.

The following shared probe settings are available:

- "Type, Name, Bandwidth" on page 151
- "Detect AutoZero, Use AutoZero" on page 151

Special features of the Rohde & Schwarz probe interface are described in these sections:

- "Probe unit, Auto attenuation" on page 153
- "Micro button action" on page 153
- "ProbeMeter" on page 154

The specific settings of Rohde & Schwarz active probes are described in the following chapters:

5.6.4.1 R&S RT-ZS single-ended probes

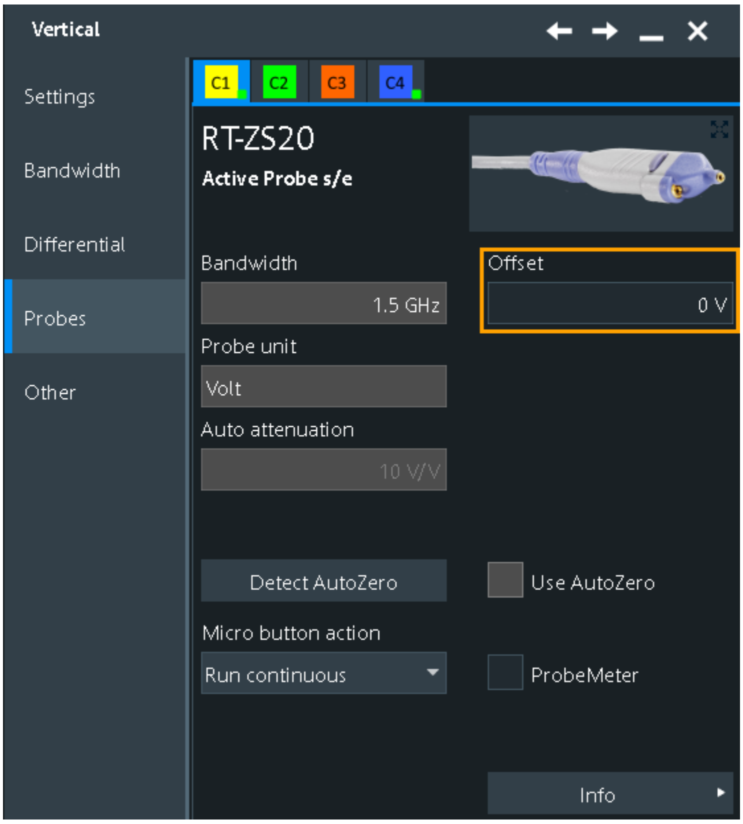


Figure 5-7: Probe setup for active single-ended probe R&S RT-ZS20

The only setting for R&S RT-ZS probes is the channel offset. See "Offset" on page 144.

5.6.4.2 R&S RT-ZD differential probes

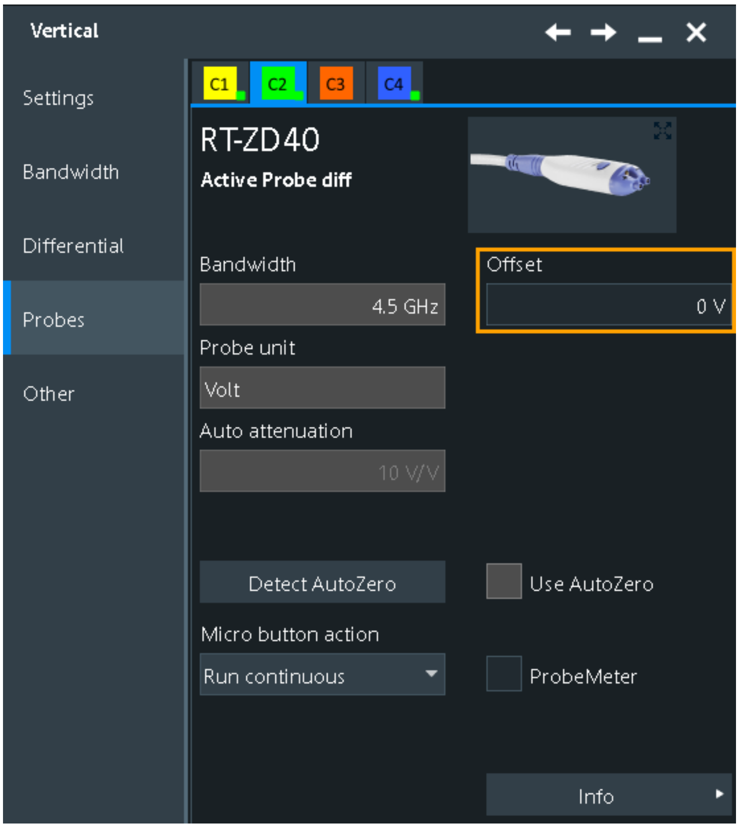


Figure 5-8: Probe setup for active differential probe R&S RT-ZD40

The offset is the differential offset. See "Offset" on page 144.

Specific settings for R&S RT-ZD probes are the following:

CM offset

Sets the common-mode offset to compensate for a common DC voltage applied to both input sockets (referenced to the ground socket). Offset compensation is particularly helpful for measurements on differential signals with high common mode levels, for example, current measurements using a shunt resistor. You can measure the common mode input voltage using the R&S ProbeMeter.

The setting is available for Rohde & Schwarz differential probes, and for modular probes in DM or CM mode (see "DM Offset, CM Offset, P Offset, N Offset" on page 164).

Remote command:
`PROBe<m>:SETup:CMOFfset` on page 1354

Probe attenuator RT-ZA15

If you use the external attenuator R&S RT-ZA15 together with one of the differential active probes R&S RT-ZD10/20/30, enable RT-ZA15 to include the external attenuation in the measurements.

Remote command:
PROBe<m>:SETup:ZAXV on page 1354

5.6.4.3 R&S RT-ZPR power rail probes

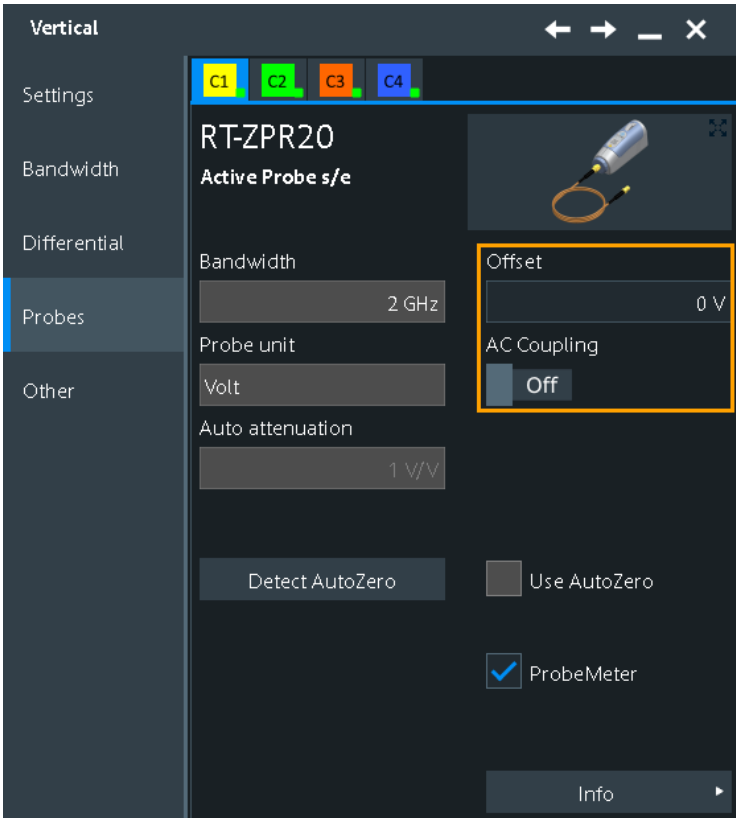


Figure 5-9: Probe setup for power rail probe R&S RT-ZPR

The offset is the channel offset. See "Offset" on page 144.

If the ProbeMeter is active, an additional function is provided in the "ProbeMeter" result box:

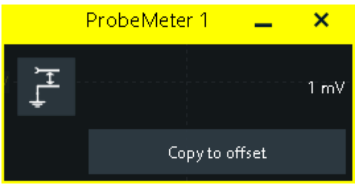


Figure 5-10: ProbeMeter result box for

Specific settings for R&S RT-ZPR probes are the following:

AC Coupling
Enables AC coupling in the R&S RT-ZPR power rail probes, which removes DC and very low-frequency components. The R&S RT-ZPR probe requires 50 Ω input termination, for which the channel AC coupling is not available. The probe setting allows AC coupling also at 50 Ω inputs.

Remote command:
PROBe<m>:SETup:ACCoupling on page 1354

Copy to offset
Sets the measured ProbeMeter value as offset. Thus, the value is considered in measurements.

Remote command:
PROBe<m>:SETup:ADVanced:PMToffset on page 1355

5.6.4.4 R&S RT-ZHD high-voltage differential probes

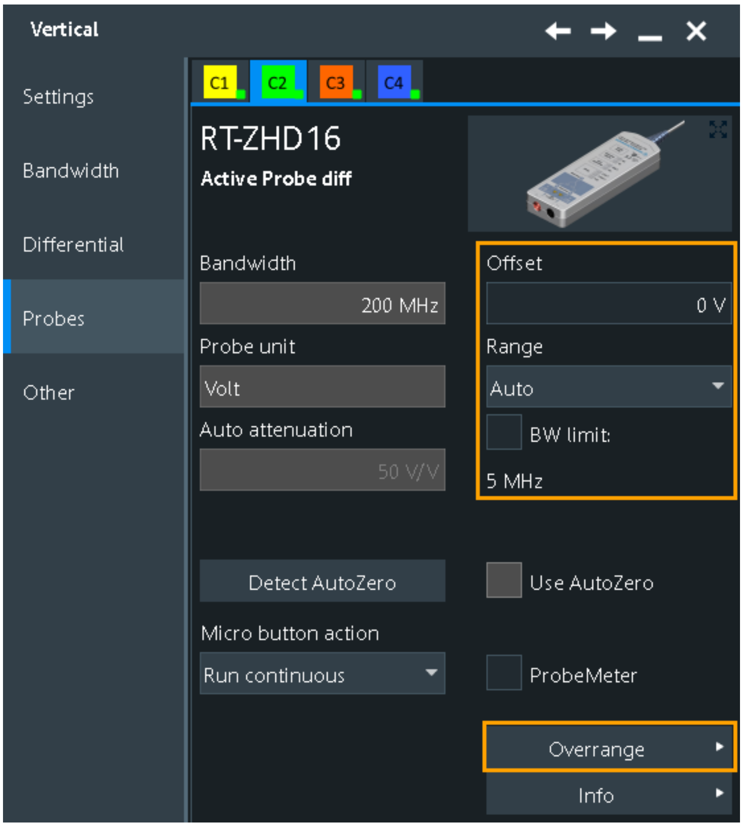


Figure 5-11: Probe setup for R&S RT-ZHD probes

The offset is the differential offset. See "Offset" on page 144.

Specific settings for R&S RT-ZHD probes are the following:

Range
Sets the voltage range of a R&S RT-ZHD probe. You can set the range on the probe control box or at the oscilloscope.

"Auto"	The voltage range is set only at the oscilloscope with "Vertical scale".
"Low"	Sets the lower voltage range of the connected probe. The selected value is shown in "Auto Attenuation".
"High"	Sets the higher voltage range of the connected probe. The selected value is shown in "Auto Attenuation".

Remote command:
`PROBe<m>:SETup:ADVanced:RANGe` on page 1355

BW limit
Activates the lowpass filter in the probe control box and displays the used limit. You can also set the filter directly on the probe control box.

Remote command:
`PROBe<m>:SETup:ADVanced:FILTer` on page 1355

Buzzer
Activates the acoustic overrange warning in the probe control box. You can also activate the sound directly on the probe control box.

Remote command:
`PROBe<m>:SETup:ADVanced:AUDioverload` on page 1356

Negative, Positive, Differential Overrange
The color turns red if the voltage exceeds the probe range. The indicators are also available on the probe control box.

5.6.5 Modular probes

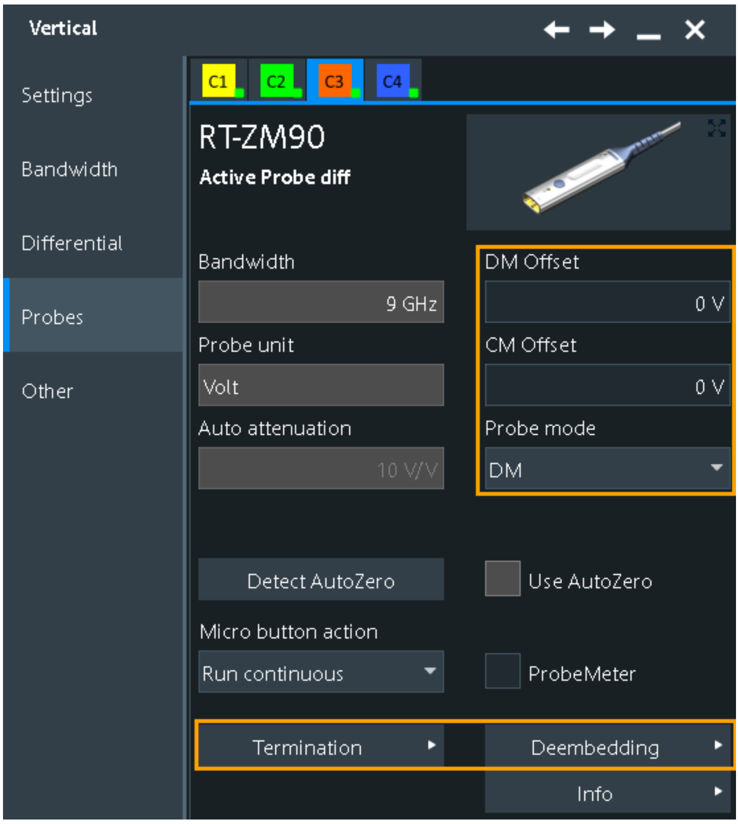
The probes of the R&S RT-ZM family are modular probes. They have a probe head and a probe amplifier connected by a cable, and various probe tip modules and tip cables for different applications. R&S RT-ZM probes are equipped with Rohde & Schwarz probe interface, and provide special features: ProbeMeter, micro button, and a wide offset compensation range.

The following shared probe settings are available:

- "Type, Name, Bandwidth" on page 151
- "Detect AutoZero, Use AutoZero" on page 151

Special features of the Rohde & Schwarz probe interface are described in these sections:

- "Probe unit, Auto attenuation" on page 153
- "Micro button action" on page 153
- "ProbeMeter" on page 154



5.6.5.1 Setup parameters of modular probes

The basic setup parameters of all modular probes are the measurement mode and the offset settings.

Access: "Menu" > "Vertical" > "Probes"

Make sure to adjust also the deembedding settings under "Probe Deembedding" (see Chapter 5.6.5.2, "Deembedding for modular probes", on page 164).

Probe mode
Sets the measurement mode of modular probes.

The modular probes of the R&S RT-ZM family have a multi-mode function. You can switch between single-ended, differential and common mode measurements without reconnecting or resoldering the probe. You can set the probe mode in the dialog box, and you can assign the probe mode setting to the micro button.

If you use the R&S RT-ZMA30 browser module, only DM measurements are possible because this module has no ground connector.

The measurement modes are:

"DM" Differential mode input voltage (V_{dm}), the voltage between the positive and negative input terminal.

$$V_{dm} = V_p - V_n$$

"CM" Common mode input voltage (V_{cm}), the mean voltage between the positive and negative input terminal vs. ground.

$$V_{cm} = \frac{V_p + V_n}{2}$$

"P" Positive single-ended input voltage (V_p). The voltage between the positive input terminal and ground.

"N" Negative single-ended input voltage (V_N). The voltage between the negative input terminal and ground.

Remote command:
PROBe<m>:SETup:PRMode on page 1356

DM Offset, CM Offset, P Offset, N Offset
Compensate offset voltages. Available offsets depend on the selected probe mode.

The offset of the selected probe mode is used as channel offset and considered automatically for correction. For example, in CM mode, the common mode offset is used as channel offset. See also: "Offset" on page 144.

"DM Offset" Compensates a DC voltage applied between the positive (V_p) and the negative (V_n) input terminal at the probe tip.

"CM Offset" Compensates a DC voltage applied to both input terminals referenced to ground. See also: "CM offset" on page 159.

"P Offset" Compensates a DC voltage applied to the positive input terminal (V_p) referenced to ground.

"N Offset" Compensates a DC voltage applied to the negative input terminal (V_n) referenced to ground.

Remote command:
PROBe<m>:SETup:DMOFfset on page 1357
PROBe<m>:SETup:CMOFfset on page 1354
PROBe<m>:SETup:NOFFset on page 1357
PROBe<m>:SETup:POFFset on page 1358

5.6.5.2 Deembedding for modular probes

Deembedding removes the parasitic effects of the measurement setup from the measured signal. These effects are typically increasing when signal frequency increases. Thus, deembedding is useful or even necessary when measuring signals of 4 GHz frequency or higher.

Access: "Vertical" menu > "Probe Setup" > "Probe Deembedding"

Select probe tip module, Pin spacing
Selects the tip module that is used for measurement.

If R&S RT-ZMA40 is selected, choose also the used submodule: semi-rigid cables or none.

For the browser module R&S RT-ZMA30, measure the space between the pins, and select the appropriate value.

Remote command:
`PROBe<m>:DEEMbedding:TIPModule[:SELect]` on page 1360
`PROBe<m>:DEEMbedding:TIPModule:ZMA<n>:SUBModule` on page 1361

Probing, Terminating
Sets the probing mode.

Select "Probing" if you use a high-ohmic probe and measure on an existing line in parallel to the load.

Select "Terminating" if the measuring equipment is the load of the line.

Remote command:
`PROBe<m>:DEEMbedding:MODE` on page 1362
`DEEMbedding<m>:COMPonent<n>:MODE` on page 2557

Effective bandwidth
Sets the maximum bandwidth for probe deembedding until which the signal is corrected. This maximum value is the minimum bandwidth value of the probe bandwidth, tip module bandwidth, and oscilloscope bandwidth. It cannot be higher than the highest frequency in a used S-parameter file.

Consider that most tip modules support the full bandwidth of the probe, but some tip modules have limited bandwidth.

If deembedding option R&S RTO-K121 is active, the instrument uses the effective bandwidth that is set in the "Deembedding" > "Setup" dialog box.

Remote command:
`PROBe<m>:DEEMbedding:BANDwidth` on page 1361

Probe mode
See "Probe mode" on page 163.

Remove probe load
If "Probing" is selected, you can remove the loading of the probe.

If the probe loading is removed, you see the signal that would be at the measurement point if the probe's input impedance is ideal (infinite impedance).

Without removing the loading, you see the real signal at the measurement point, including the probe loading.

Remote command:
`PROBe<m>:DEEMbedding:REMProbeload` on page 1362
`DEEMbedding<m>:COMPonent<n>:LOAD:REMProbeload` on page 2558

Source impedance
Shows the source impedance of the DUT. If an R&S RT-ZM probe is connected, the value depends on the selected probe mode: common mode, differential, or single-ended measurement.

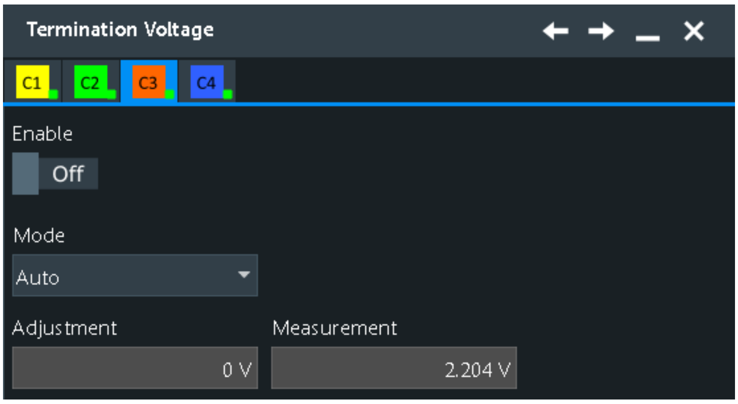
Remote command:
DEEMbedding<m>:COMPonent<n>:LOAD:IMPedance? on page 2558
PROBe<m>:DEEMbedding:LOAD:IMPedance? on page 1362

5.6.5.3 Termination voltage with R&S RT-ZMA40 SMA module

Termination voltage is relevant if you use the R&S RT-ZMA40 SMA module. The SMA module applies a termination voltage (±4 V) to the DUT to enable measurements against a common mode DC voltage instead of ground. This measurement is required for many digital signal standards.

The termination voltage can be controlled by the oscilloscope. Therefore, connect the V_T terminal of the R&S RT-ZM probe amplifier to the V_T terminal of the R&S RT-ZMA40 SMA module using the red DC lead (see R&S RT-ZM User Manual). The required termination voltage is measured and adjusted automatically, but can also be set manually.

Access: "Menu" > "Vertical" > "Probes" > "Termination"



Enable
Activates control of the termination voltage.

Remote command:
PROBe<m>:SETup:TERM:STATe on page 1358

Mode
In "Auto" mode, the instrument uses the measured common mode voltage for termination.

In "Manual" mode, you can enter the voltage to be used for termination. Use the manual mode if you know the common mode voltage of the DUT.

Remote command:
PROBe<m>:SETup:TERM:MODE on page 1358

Adjustment
Sets the voltage to be used for termination to DC voltage.

Remote command:
PROBe<m>:SETup:TERM:ADJust on page 1359

Measurement
Shows the measured common mode voltage.

Remote command:
PROBe<m>:SETup:TERM:MEASure? on page 1359

5.6.6 Setup for predefined probes

Probes that cannot be detected, but their characteristics are known to the R&S RTO are called "Predefined probes".

The following shared probe settings are available:

- "Type, Name, Bandwidth" on page 151
- "Detect AutoZero, Use AutoZero" on page 151

The probe attenuation of the selected probe is also shown in the "Probes" tab:

- "Probe unit, Auto attenuation" on page 153

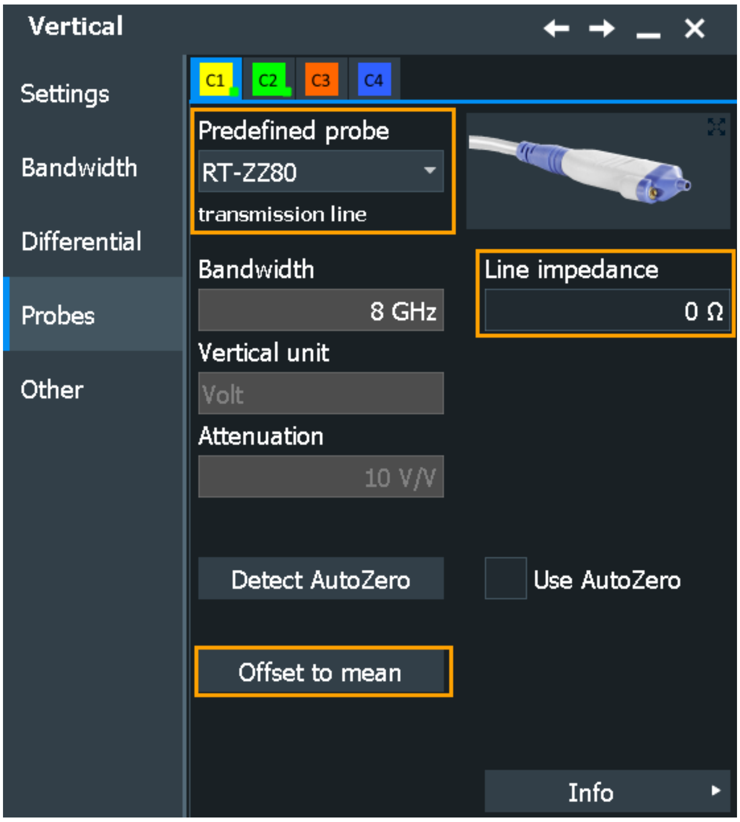


Figure 5-12: Probe setup for transmission line probe R&S RT-ZZ80

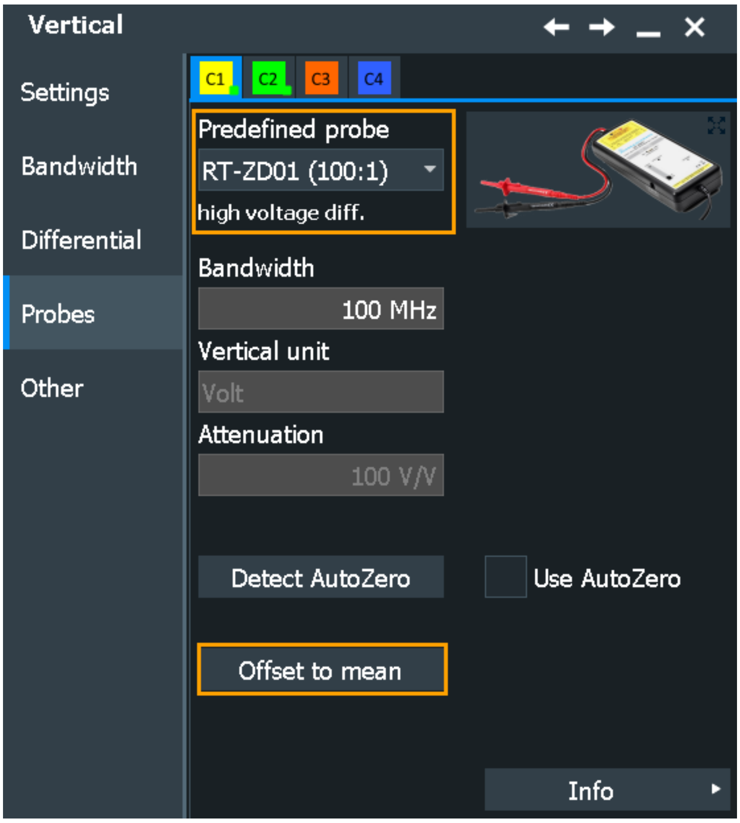


Figure 5-13: Probe setup for R&S RT-ZD01

Specific settings for predefined probes are the following:

Predefined probe

List of probes that are known to the instrument. The instrument lists only probes that match the instrument coupling and, if connected, also the used adapter.

Select the used probe on the list. The corresponding "Vertical unit" and the "Attenuation" or "Gain" are shown.

All other unrecognized probes that are not listed, are unknown probes. For these probes, set "Predefined probe" to "None". See Chapter 5.6.8, "Setup for unknown probes", on page 172.

Remote command:

`PROBe<m>:SETup:ATTenuation:DEFProbe` on page 1363

Offset to mean
Performs an automatic compensation for a DC component of the input signal using the result of a background mean measurement. The result is shown in "Offset". The function is probe-independent and supports quick and convenient measurements of input signals with different DC offsets. It detects offset values even when the signal is out of the current measurement range. It also sets the zero level to the determined DC offset in the middle of the screen and thus prevents clipping of the waveform.

Remote command:
`PROBe<m>:SETup:OFFSet:TOMean` on page 1364

Line impedance
If the transmission line probe R&S RT-ZZ80 is selected, enter the impedance of the measured line.

The actual attenuation of the transmission line probe depends on the impedance of the line Z_0:

$Attenuation = 10 + Z_0 / 100$

The instrument uses the actual attenuation to determine the measurement values.

5.6.7 Setup for current probes

The setup and adjustment of current probes depends on the output connector of the probe: BNC or Rohde & Schwarz probe box.

The following shared probe settings are available:

- "Type, Name, Bandwidth" on page 151
- "Detect AutoZero, Use AutoZero" on page 151

Current probes R&S RT-ZCxx

The current probes **R&S RT-ZCxx** have BNC connectors. They are known to the R&S RTO as predefined probes, see Chapter 5.6.6, "Setup for predefined probes", on page 167. Demagnetizing and zero adjustment is done on the probe, see the probe's User Manual for details. Make sure to demagnetize and adjust the probe before taking measurements.

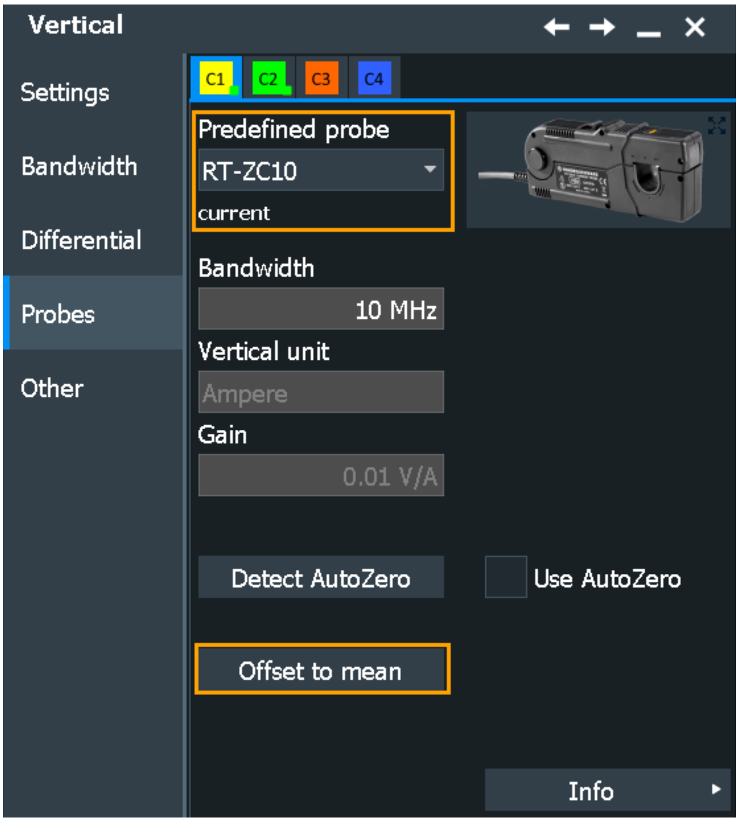


Figure 5-14: Probe setup for current probes R&S RT-ZC10

Current probes R&S RT-ZCxxB

Current probes **R&S RT-ZCxxB** have a Rohde & Schwarz probe interface; they are powered and remotely controlled by the oscilloscope.

When the probe is connected, demagnetization is performed automatically.

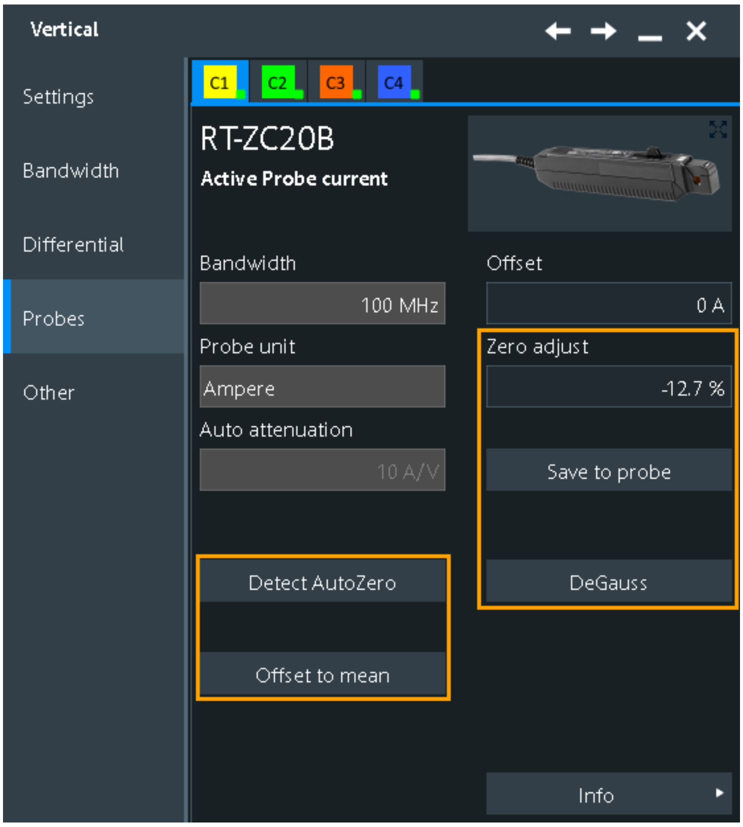


Figure 5-15: Probe setup for current probes R&S RT-ZC20B

For all current probes, attenuation or gain is shown in the "Setup" tab, and you can set the offset to mean:

- "Probe unit, Auto attenuation" on page 153
- "Offset to mean" on page 169

Current probes R&S RT-ZCxxB are adjusted by the following functions:

DeGauss

Demagnetizes the core if it has been magnetized by switching the power on and off, or by an excessive input. Always carry out demagnetizing before measurement.

The demagnetizing process takes about one second. During demagnetizing, a demagnetizing waveform is displayed.

Demagnetizing is done automatically when R&S RT-ZCxxB is connected to the oscilloscope, or when "Detect AutoZero" is performed.

Remote command:
`PROBe<m>:SETup:DEGauss` on page 1364

Detect AutoZero
If a current probe is connected, the function demagnetizes the probe's sensor head and sets the waveform to zero position to correct the error offset. Thus, it compensates for the remanence and offset caused by temperature drift.

For R&S RT-ZCxxB probes, the determined "Zero adjust" value is displayed and can be saved in the probe head.

See also "Detect AutoZero, Use AutoZero" on page 151.

Remote command:
`PROBe<m>:SETup:OFFSet:AZERo` on page 1347

Zero adjust
Zero adjust corrects the effect of an offset caused by temperature drift, and compensates for the remanence. The setting is only available if DC coupling is set.

To set the waveform to zero level by the instrument, use "Detect AutoZero". The detected value is displayed.

Alternatively, you can adjust the value manually until the waveform is set to zero level. Make sure to demagnetize the probe before zero adjustment.

The value is given in percent of the maximum range, which is internally defined. The actual setup range depends on the temperature drift, the measured current and other variables, and it can change over time. If you measure high currents, the probe core magnetizes, which impairs the measurement results. Therefore, repeat "Detect AutoZero" before the measurement.

Remote command:
`PROBe<m>:SETup:OFFSet:ZADJust` on page 1364

Save to probe
Saves the "Zero adjust" value in the probe box. If you connect the probe to another channel or to another R&S RTx oscilloscope, the value is read out again, and you can use the probe without further adjustment.

Remote command:
`PROBe<m>:SETup:OFFSet:STPRobe` on page 1364

5.6.8 Setup for unknown probes

If the R&S RTO cannot detect the probe, and the probe is not a predefined one, you can set the probe parameters manually.

The following shared probe settings are available:

- "Type, Name, Bandwidth" on page 151
- "Detect AutoZero, Use AutoZero" on page 151

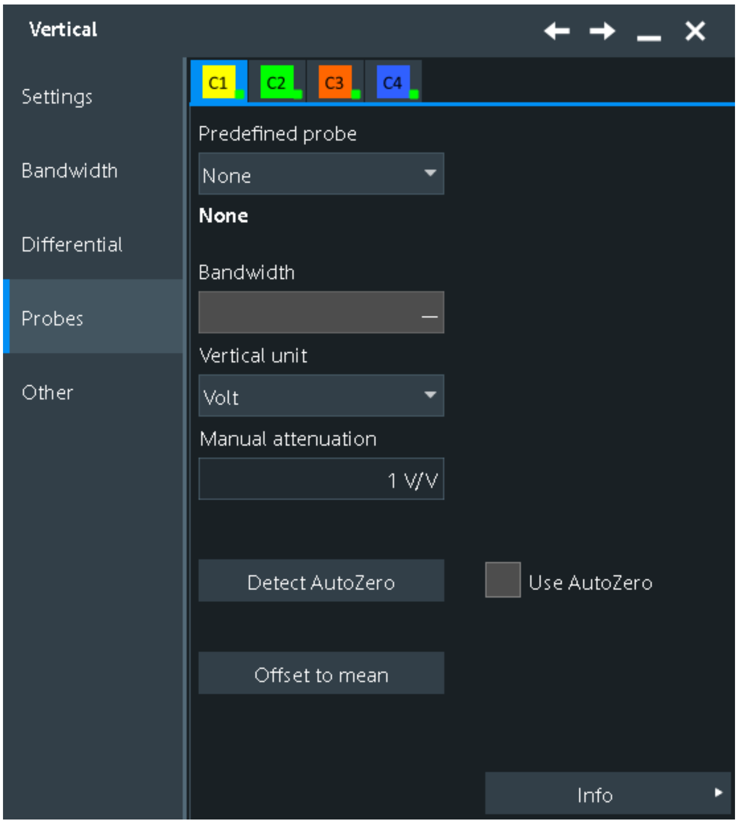


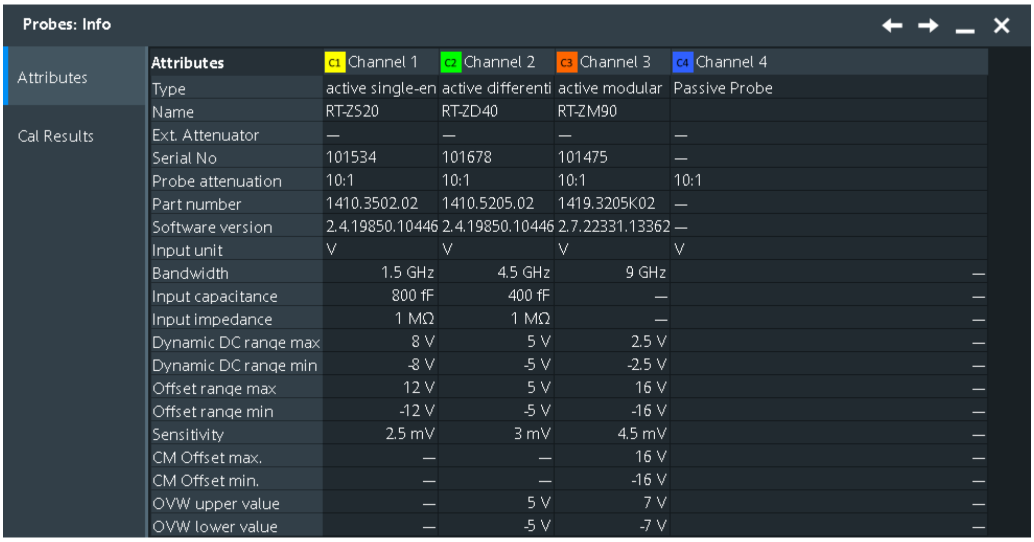
Figure 5-16: Probe setup for an unknown probe

Set the unit and the attenuation or gain of the probe: "Vertical unit, Attenuation, Gain" on page 157.

5.6.9 Probe attributes

To check the attributes of the connected probes, select "Info" in the "Vertical" > "Probes" tab.

For a specification of the probe parameters, refer to the data sheet.



Probes: Info

Attributes | Cal Results

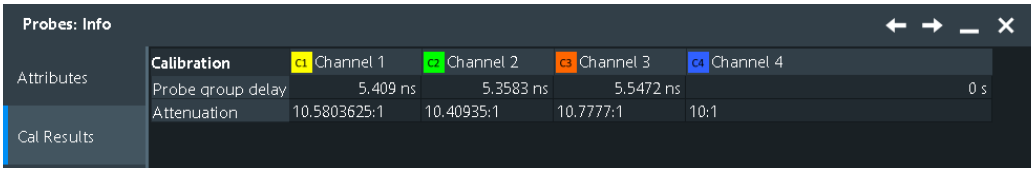
Attributes	Channel 1	Channel 2	Channel 3	Channel 4
Type	active single-en	active differenti	active modular	Passive Probe
Name	RT-ZS20	RT-ZD40	RT-ZM90	
Ext. Attenuator	—	—	—	—
Serial No	101534	101678	101475	—
Probe attenuation	10:1	10:1	10:1	10:1
Part number	1410.3502.02	1410.5205.02	1419.3205K02	—
Software version	2.4.19850.10446	2.4.19850.10446	2.7.22331.13362	—
Input unit	V	V	V	V
Bandwidth	1.5 GHz	4.5 GHz	9 GHz	—
Input capacitance	800 fF	400 fF	—	—
Input impedance	1 MΩ	1 MΩ	—	—
Dynamic DC range max	8 V	5 V	2.5 V	—
Dynamic DC range min	-8 V	-5 V	-2.5 V	—
Offset range max	12 V	5 V	16 V	—
Offset range min	-12 V	-5 V	-16 V	—
Sensitivity	2.5 mV	3 mV	4.5 mV	—
CM Offset max.	—	—	16 V	—
CM Offset min.	—	—	-16 V	—
OVW upper value	—	5 V	7 V	—
OVW lower value	—	-5 V	-7 V	—

Remote commands:

- `PROBe<m>:ID:SWVersion?` on page 1365
- `PROBe<m>:ID:PRDate?` on page 1365
- `PROBe<m>:ID:PARTnumber?` on page 1365
- `PROBe<m>:ID:SRNumber?` on page 1366
- `PROBe<m>:SETup:CAPacitance?` on page 1366
- `PROBe<m>:SETup:IMPedance?` on page 1366

5.6.10 Calibration results

To check the attributes of the connected probes, select "Info" > "Cal Results" in the "Probes" tab.



Probes: Info

Attributes | Cal Results

Calibration	Channel 1	Channel 2	Channel 3	Channel 4
Probe group delay	5.409 ns	5.3583 ns	5.5472 ns	0 s
Attenuation	10.5803625:1	10.40935:1	10.7777:1	10:1

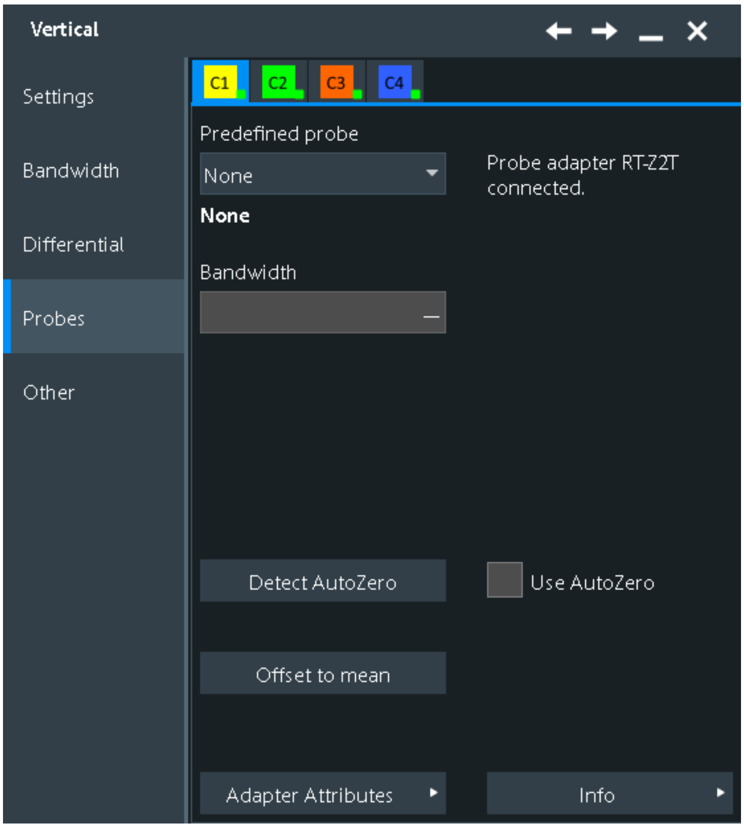
5.6.11 Probe adapter R&S RT-Z2T

Using the R&S RT-Z2T probe interface adapter, you can connect selected Tektronix active probes with TekProbe BNC™ level II interface.

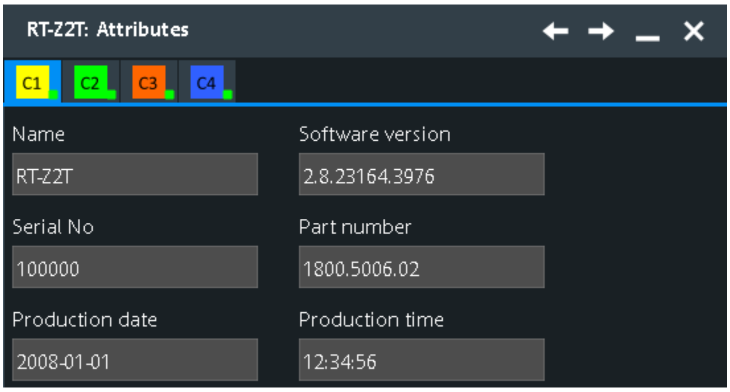
1. Connect the R&S RT-Z2T adapter to the channel input.
2. Connect the probe to the adapter.

   The instrument identifies the adapter. The R&S RTO lists the supported Tektronix probes as "Predefined probe".

3. Select "Menu" > "Vertical" > "Probes".
   The dialog shows that the R&S RT-Z2T probe interface adapter is connected.



4. To see the information on the adapter, select "Adapter Attributes".



5. Select the "Predefined probe" in the "Probes" tab.
   See also: Chapter 5.6.6, "Setup for predefined probes", on page 167.

Remote commands:
- PROBe<m>:SETup:ADAPter? on page 1367
- PROBe<m>:SETup:ATTenuation:TDEFprobe on page 1367

5.7 R&S RT-ZVC probe

With the R&S RTO and option R&S RTO-B1E, you can use the R&S RT-ZVC multi-channel power probe. It has an integrated 2- or 4-channel amperemeter and 2- or 4-channel voltmeter. The probe provides parallel measurements of analog or digital, voltage/current signals with excellent 18-bit resolution.

For more information on the R&S RT-ZVC probe, see also its user manual.

Source Channels

You can simultaneously connect a R&S RT-ZVC and a R&S RT-ZL04 to the R&S RTO, but no parallel operation on screen is possible.
You can acquire and measure the R&S RT-ZVC or R&S RT-ZL04 together with the analog input channels. They are running on the same horizontal scale.

If an amperemeter or voltmeter channel is activated, it can be displayed on the screen and used as a source for:

- Cursor measurements
- Automatic measurements, and also histogram, limit checks, longterm, track, quick measurements
- Mask testing
- Basic and advanced maths
- Reference waveform
- Trigger type "Edge"
- Search: all search events
- XY diagrams
- Data export

All features of the base unit, the R&S RTO-K18 option and R&S RTO-K19 option are supported.

Data export

You can save the data of the amperemeter and voltmeter channels to an XML, CSV, or BIN file. One channel per file can be saved. Files in BIN format can be reloaded to the R&S RTO as reference waveforms.

See also:

- Chapter 12.2.6, "Saving and loading waveform data", on page 469
- Chapter 12.2.1, "Waveform export files", on page 450

Remote commands for export to file:

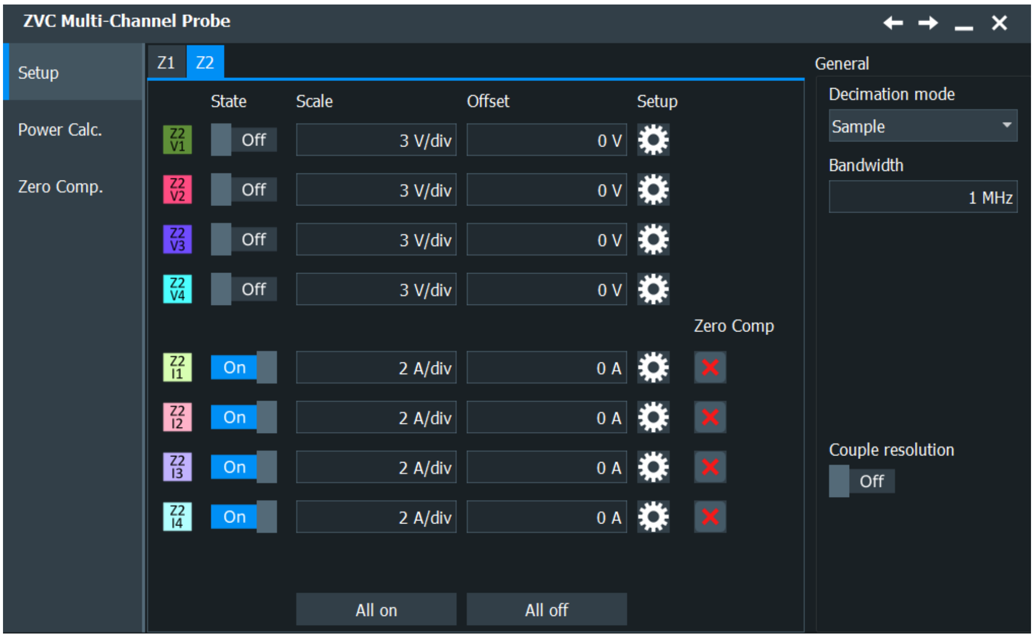
- `EXPort:WAVeform:SOURce` on page 1648
- `EXPort:WAVeform:NAME` on page 1651
- `EXPort:WAVeform:SAVE` on page 1651

5.7.1 R&S RT-ZVC overview

5.7.1.1 Setup

Access: [App Cockpit]> "Analysis" > "Multi-channel probe" > "Setup" tab

Make sure that the tab of the correct probe is selected on the left side.



State
Enables the corresponding channel of the probe. The number of available channels depend on the characteristics of your multi-channel power probe.

Remote command:
`ZVC:Z<m>:V<n>[:STATe]` on page 1377
`ZVC:Z<m>:I<n>[:STATe]` on page 1374

Scale
Defines the vertical scale for the channel in Volts per division. Increasing the scale compresses the display of the signal. Within a given operation range, modifying the scale is equivalent to scaling a display range.

For the voltmeter channels, the Scale, the Offset and the "Position" specify the operating range of the voltmeter.

Remote command:
`ZVC:Z<m>:V<n>:SCALe` on page 1376
`ZVC:Z<m>:I<n>:SCALe` on page 1372

Offset

The vertical center of the selected channel is shifted by the offset value and the signal is repositioned within the diagram area. Negative offset values move up the waveform, positive values move it down.

Within a given operation range, modifying the offset is equivalent to moving vertically the display range. The offset can only be modified such that the display range reaches at most the limits of the operation range.

For the voltmeter channels, the Scale, the Offset and the "Position" specify the operating range of the voltmeter.

Remote command:
`ZVC:Z<m>:V<n>:OFFSet` on page 1375
`ZVC:Z<m>:I<n>:OFFSet` on page 1370

Setup

Opens the voltage or current settings for the selected channel. See Chapter 5.7.2, "ZVC voltage settings", on page 181 and Chapter 5.7.3, "ZVC current settings", on page 182.

Zero Comp

Shows the status of the zero compensation of each current channel. The green checkmark indicates that the zero offset is compensated automatically. To adjust the settings, tap the icon. For details, see Chapter 5.7.1.3, "Zero compensation", on page 180.

Remote command:
`ZVC:Z<m>:I<n>:ZERComp:STATe?` on page 1379

All on

Enables all available channels.

All off

Disables all available channels.

Decimation mode

Selects the decimation mode for all R&S RT-ZVC probes. Decimation reduces the data stream of the ADC to a stream of waveform points with lower sample rate and a less precise time resolution.

"Sample"	One of n samples in a sample interval of the ADC is recorded as waveform point, the other samples are discarded. The time between the two adjacent waveform points is exactly the resolution. Very short glitches might remain undiscovered by this method.
"Peak detect"	The minimum and the maximum of n samples in a sample interval are recorded as waveform points, the other samples are discarded.
"High res"	The average of n sample points is recorded as one waveform sample. Averaging reduces the noise, the result is a more precise waveform with higher vertical resolution. The high measurement resolution is suitable for high accuracy measurements of instantaneous values.

Remote command:
`ZVC:TYPE` on page 1369

Bandwidth
Sets the bandwidth limit of all R&S RT-ZVC probes. The bandwidth specifies the maximum frequency at which a purely sinusoidal signal is still transferred at 89 % (1 dB) of its amplitude.

The bandwidth of some current channels is restricted to 300 KHz due to their vertical settings.

Remote command:
ZVC:BANDwidth on page 1369

Couple resolution
Sets the resolution of all R&S RT-ZVC channels.

"On" The resolution of the analog channels is applied to R&S RT-ZVC channels. The signal is automatically interpolated or decimated to get the analog resolution.

"Off" The resolution of R&S RT-ZVC channels is set in a way so that the record length of the waveforms is minimum 1000 samples.

Remote command:
ZVC:RESCoupled on page 1370

5.7.1.2 Power calculation

Access: [App Cockpit]> "Analysis" > "Multi-channel probe" > "Power Cal." tab

State
Enables the corresponding channel of the probe. The number of available channels depend on the characteristics of your multi-channel power probe.

Remote command:
ZVC:Z<m>:V<n>[:STATe] on page 1377
ZVC:Z<m>:I<n>[:STATe] on page 1374

Measurement Impedance
Sets the impedance of the probe channel for power calculations and measurements.

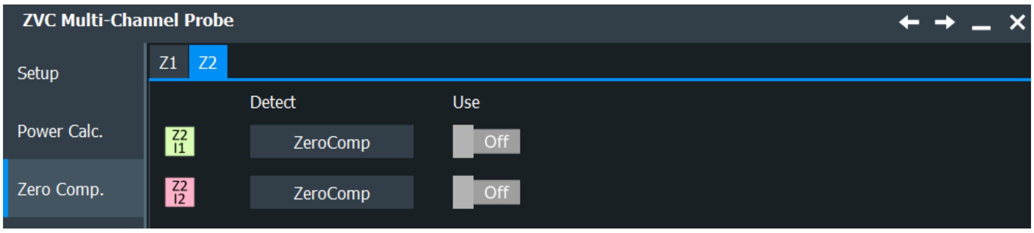
Remote command:
`ZVC:Z<m>:V<n>:IMPedance` on page 1375
`ZVC:Z<m>:I<n>:IMPedance` on page 1370

5.7.1.3 Zero compensation

Access: [App Cockpit]> "Analysis" > "Multi-channel probe"> "ZeroComp" tab

Zero compensation is used to avoid negative currents in measurement results, and to improve the measurement accuracy. The determined compensation is valid as long as the temperature is constant and the probe settings are unchanged.

1. Disconnect the DUT from the power supply.
2. Adjust the current settings, in particular, the shunt settings. See Chapter 5.7.3, "ZVC current settings", on page 182.
3. Connect one probe tip of the R&S RT-ZVC probe to the high voltage pin of the DUT. Disconnect the other probe tip from the DUT.
4. Open the "ZeroComp" settings: [App Cockpit]> "Analysis" > "Multi-channel probe"> "ZeroComp" tab
5. Tap "ZeroComp" to determine the zero offset.
6. Enable "Use" to compensate for the zero offset.



ZeroComp
Measures the zero offset, the mean value on a currentless DUT. If temperature changes, or if you change the shunt or other probe settings, repeat the measurement.

Remote command:
`ZVC:Z<m>:I<n>:ZERComp:DETect` on page 1378

Use
If enabled, the measured "ZeroComp" offset value is compensated automatically.

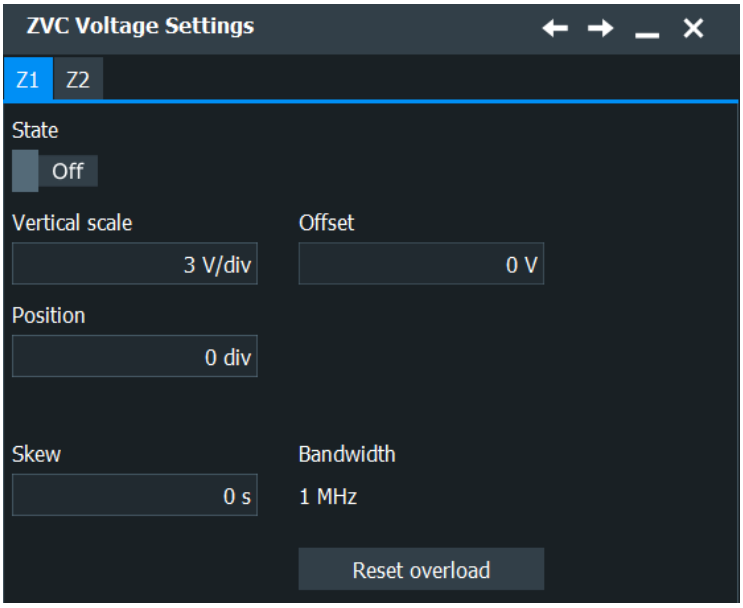
Remote command:
`ZVC:Z<m>:I<n>:ZERComp:USE` on page 1379

5.7.2 ZVC voltage settings

Access: [App Cockpit]> "Analysis" > "Multi-channel probe"> "Setup" tab > "Setup" icon of voltage channel

See also:

- Scale
- Offset



Channel
Selects the voltmeter channel to be configured.

State
Enables the corresponding voltage channel of the probe. The number of available channels depend on the characteristics of your multi-channel power probe.

Remote command:
`ZVC:Z<m>:V<n>[:STATe]` on page 1377

Position
Moves the selected signal up or down in the diagram. The visual effect is the same as for Offset. While the offset sets a voltage, position is a graphical setting given in divisions. Within a given operation range, modifying the position is equivalent to moving vertically the display range. The position can only be modified such that the display range reaches at most the limits of the operation range.

For the voltmeter channels, the Scale, the Offset and the "Position" specify the operating range of the voltmeter.

Remote command:
`ZVC:Z<m>:V<n>:POSition` on page 1376

Bandwidth
Displays the bandwidth of the current channel. You can set the probe bandwidth in the "Setup" dialog.

Remote command:
`ZVC:Z<m>:V<n>:BANDwidth?` on page 1375

Skew
Sets the skew, a delay value, that is known from the circuit specifics but cannot be compensated by the instrument automatically.

Remote command:
`ZVC:Z<m>:V<n>:SKEW` on page 1377

Reset Overload
Resets the overload indication at the probe.

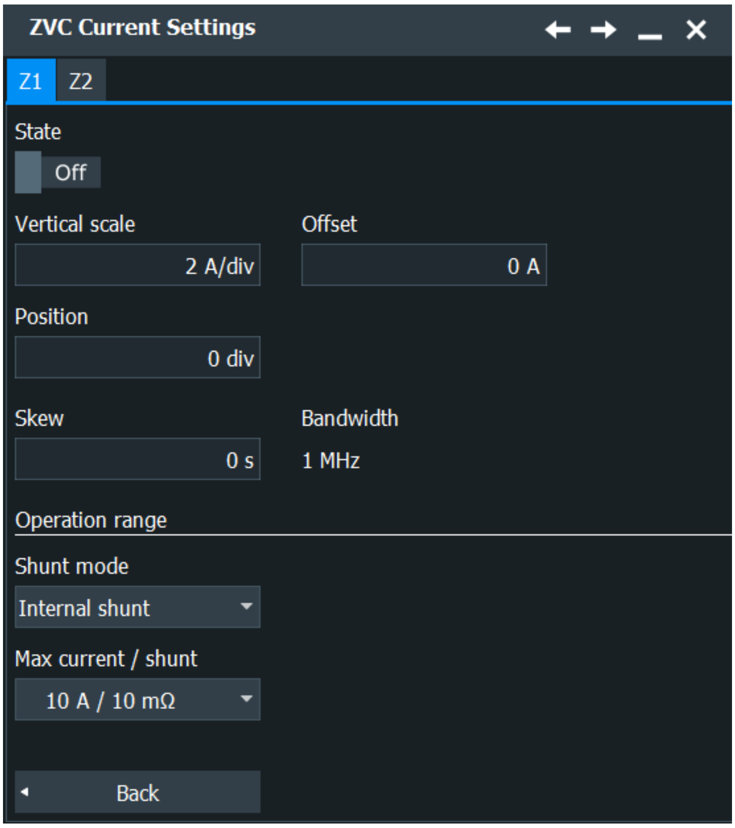
Remote command:
`ZVC:Z<m>:V<n>:OVERload:RSTO` on page 1376

5.7.3 ZVC current settings

Access: [App Cockpit]> "Analysis" > "Multi-channel probe"> "Setup" tab > "Setup" icon of current channel

See also:

- Scale
- Offset



Channel
Selects the amperemeter channel to be configured.

State
Enables the corresponding current channel of the probe. The number of available channels depend on the characteristics of your multi-channel power probe.

Remote command:
`ZVC:Z<m>:I<n>[:STATe]` on page 1374

Bandwidth
Displays the bandwidth of the current channel. You can set the probe bandwidth in the "Setup" dialog.

Remote command:
`ZVC:Z<m>:I<n>:BANDwidth?` on page 1369

Position
Moves the selected signal up or down in the diagram. The visual effect is the same as for Offset. While the offset sets a current, position is a graphical setting given in divisions. Within a given operation range, modifying the position is equivalent to moving vertically the display range. The position can only be modified such that the display range reaches at most the limits of the operation range.

Remote command:
`ZVC:Z<m>:I<n>:POSition` on page 1371

Skew
Sets the skew, a delay value, that is known from the circuit specifics but cannot be compensated by the instrument automatically.

Remote command:
`ZVC:Z<m>:I<n>:SKEW` on page 1374

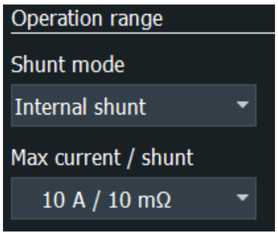
Shunt mode
Selects the internal or external shunt mode.

Regarding the shunt selection, i.e. the burden voltage level, there is a trade-off between the burden of the circuit under test and the SNR at the frontend input. The burden voltage is the DUT circuit loading caused by leads, connectors and the amperemeter circuit.

From the DUT perspective, the burden voltage has to be kept low not to distort the device operation. In contrast, from the probe's view the voltage has to be as large as possible to obtain a good SNR. For that reason, the external shunt can be applied to get the best compromise of both for a specific measurement range.

Remote command:
`ZVC:Z<m>:I<n>:SHUNt:MODE` on page 1372

Internal Shunt Mode
If Shunt mode is set to "Internal shunt", include the settings for the internal shunt mode.



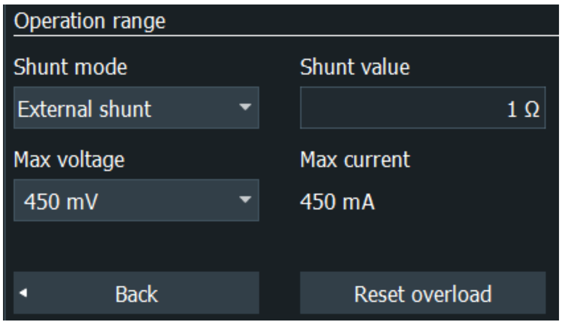
Max current / shunt ← Internal Shunt Mode
Selects the maximum current and the internal shunt value.

With the maximum current and the internal shunt selection, the operating range of the amperemeter is specified. At the same time, the burden voltage at the amperemeter input can be estimated. For values of the total round-trip resistance that can be seen at the test lead ends, consider the data sheet.

For using internal shunts, the circuit under test needs to be interrupted so that the current can flow through the probe.

Remote command:
`ZVC:Z<m>:I<n>:SHUNt:MAXCurrent` on page 1373

External shunt mode
If Shunt mode is set to "External shunt", include the settings for the external shunt mode.



Max voltage ← External shunt mode
Selects the maximum voltage at the external shunt.

Remote command:
ZVC:Z<m>:I<n>:SHUNt:MAXVoltage on page 1374

Shunt value ← External shunt mode
Sets the shunt value of the external shunt resistor.

Remote command:
ZVC:Z<m>:I<n>:SHUNt:EVALue on page 1372

Max current ← External shunt mode
Displays the maximum current.

Remote command:
ZVC:Z<m>:I<n>:SHUNt:MXCValue? on page 1373

Reset Overload ← External shunt mode
Resets the overload indication at the probe.

Remote command:
ZVC:Z<m>:I<n>:OVERload:RSTO on page 1371

5.8 Differential signals

With R&S RTO, you can easily analyze differential signals using single-ended probes, or even cables. The instrument processes the input of single-ended probes on analog channels and creates the differential and common mode waveforms. Similar to Rohde & Schwarz modular probes, you can display the waveforms of differential voltage, common mode voltage, positive single-ended voltage and negative single-ended voltage. Triggering on differential signals is also possible.

Deembedding is possible if option R&S RTO-K121 is installed.

Deembedding is applied to the two single-ended captured signals, resulting in deembedded differential waveform, common mode waveform, positive single-ended waveform and negative single-ended waveform. Deembedding also considers cross-couplings.

5.8.1 Settings of differential signals

Access: "Menu" > "Vertical" > "Differential"

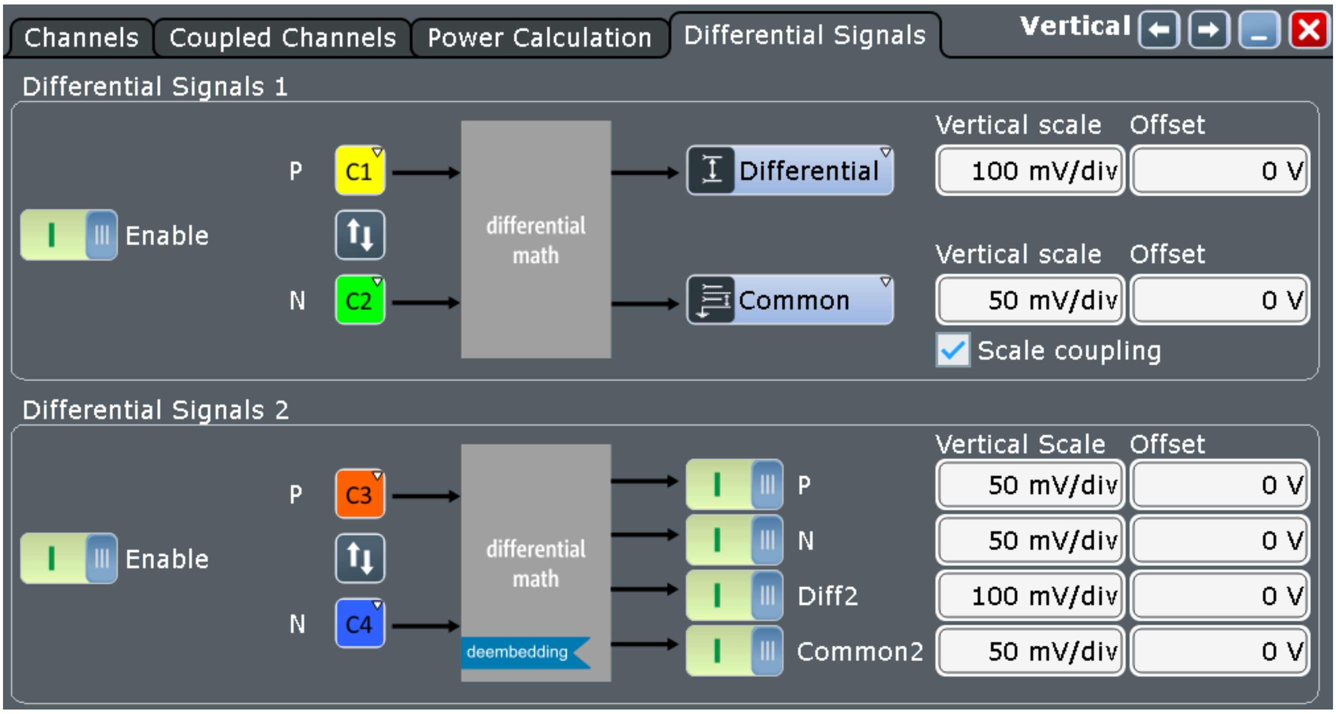


Figure 5-17: Setup of differential signals, signal 2 is deembedded

Enable
Activates the differential signal.

Remote command:
`DIFFerential<m>:STATe` on page 1382

P, N
Select the analog input channel of the positive and negative signal.

For differential signal 1, analog Ch1 and Ch2 are used. Differential signal 2 uses Ch3 and Ch4. You can switch the channels with the arrow button.

Remote command:
`DIFFerential<m>:PSIGnal[:SELect]` on page 1382
`DIFFerential<m>:NSIGnal[:SELect]` on page 1382

Output: P, N, Differential, Common
Select the waveforms that are displayed as result of differential processing.

The number of output waveforms per differential signal depends on the deembedding:
- No deembedding: 2 output waveforms.
- With deembedding: up to 4 output waveforms. All output waveforms can be displayed at the same time.

In the diagram, you see the output waveforms, and the signal icons of the input channels are grayed out. The input channels are only visible if "P" or "N" are selected as output.

"P", "N"
: Displays the waveform of the positive or negative input signal. Without deembedding, the input channel is shown directly, no additional waveform is created. If differential deembedding is active, you always see the deembedded waveforms, but never the captured input signal.

"Differential"
: Displays the differential signal as a new waveform.

"Common"
: Displays the common mode signal as a new waveform.

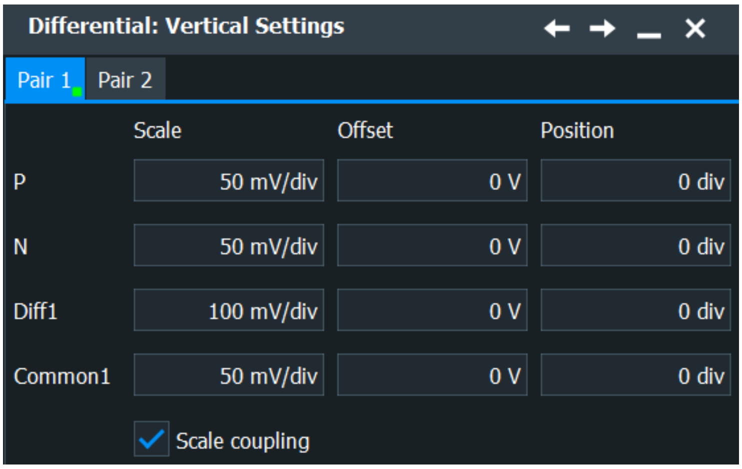
Remote command:
`DIFFerential<m>:AOUTput` on page 1382
`DIFFerential<m>:BOUTput` on page 1382
If deembedding by software is active, use the following commands:
`DIFFerential<m>:COMMon:STATe` on page 1383
`DIFFerential<m>:DIFFerential:STATe` on page 1383
`DIFFerential<m>:NSIGnal:STATe` on page 1383
`DIFFerential<m>:PSIGnal:STATe` on page 1383

Vertical scale, Offset
Vertical scale and offset of differential and common mode waveforms can be set directly in the differential setup, or in the "Vertical Setup" of differential signals. See: "Scale" on page 187 and "Offset" on page 188.

Vertical settings of P and N output are the vertical settings of the input channels.

5.8.2 Vertical setup of differential signals

Access: "Menu" > "Vertical" > "Differential" > "Vertical Setup"



Scale
Sets the vertical scale of differential and common mode waveforms.

Vertical settings of P and N output are the vertical settings of the input channels.

Remote command:
`DIFFerential<m>:COMMon:SCALe` on page 1383
`DIFFerential<m>:DIFFerential:SCALe` on page 1383

Scale coupling
If enabled, the vertical scales of P, N, differential and common mode waveforms are coupled. The scales are related as follows:

$VertScale_P = VertScale_N = VertScale_{CM} = VertScale_{Diff}/2$

You can disable the scale coupling and set an individual scale for each waveform.

Remote command:
`DIFFerential<m>:COUPling` on page 1383

Offset
Sets the offset of differential and common mode waveforms.

Vertical settings of P and N output are the vertical settings of the input channels.

Remote command:
`DIFFerential<m>:COMMon:OFFSet` on page 1384
`DIFFerential<m>:DIFFerential:OFFSet` on page 1384

Position
Sets the vertical position of differential and common mode waveforms.

Vertical settings of P and N output are the vertical settings of the input channels.

Remote command:
`DIFFerential<m>:COMMon:POSition` on page 1384
`DIFFerential<m>:DIFFerential:POSition` on page 1384

5.8.3 Deembedding of differential signals

To remove the parasitic effects of the measurement setup from the measured signal, you can use deembedding for differential signals. Deembedding requires option R&S RTO-K121.

The deembedding of differential signals is represented by one or more 4-port input components. You can activate deskewing, and set a skew time.

See also: Chapter 20, "Deembedding (Option R&S RTO-K121)", on page 1197.

1. On the "Vertical" menu, select "Differential Signals".
2. Select the input channels of the positive and negative signal.
3. Enable the differential signal.
4. Select "Deembedding".
5. Select the channel subtab of the positive input.
6. On the "Setup" tab, enable deembedding for the channel.
7. On the "Input" component, select "Configure".

8. If necessary, activate "Use skew offset". Use the deskewing if the P and N waveforms are not correctly aligned.

When you disable a deembedded differential signal in "Vertical" > "Differential Signals", deembedding is also disabled. When you enable the differential signal again, repeat the deembedding and output settings.

5.8.4 Analysis of differential signals

Before analyzing the output waveforms of differential signals, make sure that the P and N signals are correctly aligned. Without deembedding, use "Horizontal" > "Skew" to deskew the input waveforms if necessary. For deembedded waveforms, the skew setting is in the deembedding configuration.

For all output waveforms of differential signals (differential, common mode, P, N), the usual analysis methods are available:

- Trigger on differential, common mode, P, and N waveforms.
  If the differential signal is enabled ("Enable" = on), all possible output waveforms are available as trigger source, even if they are not selected and not displayed.
- Zoom
- Cursor measurements
- Automatic measurements
- Histogram
- Mathematics. In formulas, use DIFF1, DIFF2, COMMON1, COMMON2.
- FFT, spectrogram
- Mask test
- Export of waveform data. You can export the differential and common waveforms if they are selected as output and displayed. Raw data is not available.
  Without deembedding, P and N waveforms can be exported if they are selected as output and displayed (source = input channel). With software deembedding, only differential and common waveforms are available for export.

5.9 Setting up the waveform

This chapter contains the fundamental procedures for setting up the acquisition and adjusting the channel waveforms.

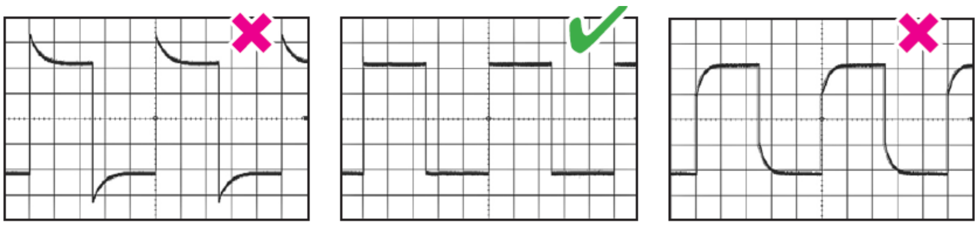
5.9.1 Adjusting passive probes

When using a passive probe, you have to compensate it when you connect it to the instrument the first time. Compensation matches the probe cable capacitance to the oscilloscope input capacitance to assure good amplitude accuracy from DC to upper bandwidth limit frequencies. A poorly compensated probe reduces the performance of the probe-oscilloscope system and introduces measurement errors resulting in distorted waveforms and inaccurate results.

Two connector pins are located on the front panel. The right pin is on ground level. The left pin supplies a square wave signal with 1 kHz for low frequency probe compensation.

1. Connect the BNC connector of the probe to input [C1].
2. Connect the probe's ground connector to the right compensation pin, and the tip with the left pin.
3. Press [Autoset].

   A square wave appears on the display.
4. Adjust the compensation trimmer of the probe to optimum square wave response. For details, refer to the documentation of your probe.



5.9.2 Setting up the signal input with autoset

Autoset is the solution for the major part of routine test-setup. It is also a good start if you need to use more complex trigger settings. Autoset finds appropriate horizontal and vertical scales, vertical offset, and trigger conditions to present a stable waveform.

1. Connect the probe to the input connector [Ch <n>].

   The instrument recognizes the probe and turns on the channel.
2. Press the [Autoset] key.

5.9.3 Adjusting the signal input manually

1. Connect the probe to the input connector [Ch <n>].

   The instrument recognizes the probe and turns on the channel.
2. Open "Menu" > "Horizontal" > "Setup" tab.

3. Set the "Time scale".
4. If you want to analyze the signal some time before or after the trigger, use the "Position" and "Reference point" to adjust the visible section of the waveform.
5. Press the channel button corresponding to the input channel. It is illuminated with the color of the channel waveform.
6. Open the "Menu" > "Vertical" > "Setup" tab.
7. Select the "Coupling".
8. Adjust the vertical "Scale", and the vertical "Position".
9. Proceed with the acquisition setup, see Chapter 5.9.4, "Setting the acquisition", on page 191.

5.9.4 Setting the acquisition

Prerequisites:

- Probes are connected.
- Vertical and horizontal settings are adjusted.

The settings are described in Chapter 5.4.2, "Mode settings", on page 134.

1. Open "Menu" > "Acquire" > "Mode" tab.
2. To configure the waveform-specific acquisition settings, enable "Multi waveform". Activate the waveforms "W1"/"W2"/"W3" you need.
   You can set up and display up to three waveforms per channel.
3. Select the "Mode" - for example, Peak detect or High res.
4. Select the "Wfm Arithmetic" - for example, Average or Envelope.
   The instrument precludes incompatible combinations, like "Peak detect" with "Average".
5. If "Wfm Arithmetic"="Average" is selected for a waveform, enter the "Average count", that is the number of waveforms used for average calculation.
6. Set the aritmetic reset condition for the average and envelope calculation:
   a) If "Arithmetic Reset > Mode"="Time" is selected, enter the "Time".
   b) If "Arithmetic Reset > Mode"="Waveforms" is selected, enter the "Count".

5.9.5 Starting and stopping acquisition

You can control the acquisition in two ways:

- Running continuous acquisition until you stop it.
- Running one acquisition or a given number of acquisitions.

If "Envelope" or "Average" is selected in the "Acquisition" tab, one acquisition means a cycle containing as many acquired waveforms as required to satisfy the reset conditions.

Prerequisites:

- Probes are connected.
- Vertical and horizontal settings are adjusted.
- Triggering is set.
- Channels to be acquired are turned on.

To start and stop continuous acquisition

1. Check if the trigger mode is set to "Norm". The trigger mode is shown in the trigger label in the upper right corner of the screen.
   If not, press the trigger [Mode] key on the front panel to toggle the setting.
2. Press the [Run Stop] key to start acquisition.
   The acquisition starts if a trigger occurs.
3. To stop , press the [Run Stop] key again.
   The acquisition stops immediately.

To acquire a limited number of acquisitions

1. Check if the trigger mode is set to "Norm". The trigger mode is shown in the trigger label in the upper right corner of the screen.
   If not, press the trigger [Mode] key on the front panel to toggle the setting.
2. Open "Menu" > "Acquire" > "Mode" tab.
3. Enter the number of acquisitions in the "Average count" field.
4. Press the [Single] key on the front panel.
   You can stop the running acquisition before it is finished by pressing the key again.

5.9.6 Using the roll mode

The roll mode can be used if the acquisition process is slow - that is if the time scale is large. In roll mode, the instrument shows the waveform immediately and saves waiting for the waveform display. The roll mode can be activated by the instrument if several conditions are fulfilled.

To set the roll mode manually

1. Make sure that all requirements for the roll mode are fulfilled: see Chapter 5.2, "Horizontal", on page 125.
2. Open "Menu" > "Horizontal" > "Roll" tab.
3. Set "Mode" to "Auto".

4. Set the "Start roll time", the acquisition time at which the instrument starts the roll mode.

5.9.7 Using fast segmentation

Fast Segmentation reduces the dead time between two waveform acquisition cycles.

The settings are described in Chapter 5.4.3, "Segmented settings", on page 137.

1. Open "Menu" > "Acquire" > "Segmented" tab.
2. Tap "Fast segmentation" to activate the fast segmentation mode.
3. If you want to sample the maximum number of acquisitions in a series, enable "Acquire maximum".
   If you want to capture a defined number of acquisitions, disable "Acquire maximum" and enter the "Required" number of acquisitions.
4. Set the "Display time".

5.9.8 Using digital filters

Before using digital filters, you determine if you want to filter input channels only or if the trigger signal is filtered too. The filter settings depend on this decision.

For details on filter settings and dependencies, see Chapter 5.5.5, "Digital filter setup", on page 148.

To filter the input channels only

1. Open "Menu" > "Vertical" > "Bandwidth" tab > "Digital Filter".
2. Set the "Trigger BW limit" to "Off".
3. Enter the "Channel BW limit" frequency for each filter.
4. Enable Filter on channel: "On | Off" for each channel to be filtered.

To filter the trigger signal

1. Open "Menu" > "Vertical" > "Bandwidth" tab > "Digital Filter".
2. Set the "Trigger BW limit" to "RF Reject".
3. Set the frequency limit for the filter: "Channel BW limit".
4. To filter the input channels too, enable Filter on channel: "On | Off" for each channel to be filtered.
   The trigger filter settings are applied also to these input channels.

6 Triggers

6.1 Basics of triggering

Triggering means to capture the interesting part of the relevant waveforms. Choosing the right trigger type and configuring all trigger settings correctly allows you to detect various incidents in analog, digital, and protocol signals.

Trigger

A trigger occurs if the complete set of trigger conditions is fulfilled. The trigger is the determining point in the waveform record. The instrument acquires continuously and keeps the sample points to fill the pre-trigger part of the waveform record. When the trigger occurs, the instrument continues acquisition until the post-trigger part of the waveform record is filled. Then it stops acquiring and waits for the next trigger. When a trigger is recognized, the instrument does not accept another trigger until the acquisition is complete and the holdoff time has expired.

Trigger setup

A simple trigger setup includes:

- Source of the trigger signal, its coupling and filtering
- Trigger type selection and setup
- Horizontal position of the trigger: see: Chapter 5.1.3.2, "Horizontal position", on page 120
- Trigger mode

The R&S RTO provides various trigger types for troubleshooting and signal analysis, for example, edge trigger, glitch trigger, interval trigger, pattern trigger, and much more.

For complex tasks like verifying and debugging designs, advanced trigger settings are available:

- Hysteresis, that is the rejection of noise to avoid unwanted trigger events caused by noise
- Holdoff to define exactly which trigger event causes the trigger
- Qualification to consider the states of digital signals on other input channels and their logical combination
- Trigger sequences to combine two trigger type conditions

Action on trigger

A trigger can initiate one or more actions, for example, saving a screenshot or saving waveform data. All available actions can be initiated at the same time.

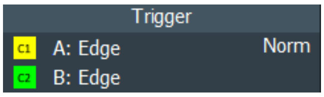
Trigger sequence

A complex trigger sequence joins two or more separate trigger conditions with an optional delay time and an optional reset time or reset condition. Similar setups are also known as multi-step trigger or A/B trigger.

Trigger information

Information on the most important trigger settings is shown in the trigger label on top of the signal bar. If you double-tap the trigger label, the "Trigger" dialog box opens. The label shows:

- Trigger mode
- Trigger type, edge/polarity and trigger source, for A- and B-trigger



6.2 Setting up the trigger

This chapter provides step-by-step procedures for the important stages of trigger setup.

6.2.1 Configuring a simple trigger

Prerequisites:

- Horizontal and vertical settings are set appropriately to the signals.
- The acquisition is running, the [Run Stop] key lights green.

For details on settings, see Chapter 6.3, "Trigger types", on page 197.

Proceed as follows:

1. Open the "Menu">"Trigger" dialog.

   The "Trigger" dialog box opens with the "Setup" tab.
2. Tap the "Source" button and select the trigger source.
3. Tap the "Type" button and select the trigger type.
4. Under "Trigger type dependent settings", configure the settings for the selected trigger type.
   See: Chapter 6.3, "Trigger types", on page 197
5. To set the trigger level automatically, tap "Find level".
6. Set the normal trigger mode. Do one of the following:
   - Press the [Mode] key on the front panel until "Normal" is shown in the trigger label.

- In the "Mode / Holdoff" tab, set the "Trigger mode" to "Normal".

6.2.2 Positioning the trigger

By positioning the trigger on the time axis, you define which part of the waveform is displayed: mainly the pre-trigger part, or the post-trigger part, or the part around the trigger point.

For details on position settings, see Chapter 5.4, "Acquire settings", on page 131.

1. Open the "Menu"> "Horizontal" dialog.
2. Set the "Reference point" and the "Position".
   If you want to set the trigger position outside the waveform display, make sure that "Restrict horizontal position to acquisition range" is disabled.

6.2.3 Using holdoff

For details on holdoff settings, see Chapter 6.4, "Mode / Holdoff", on page 223.

1. Open the "Menu">"Trigger" dialog.
2. Select the "Mode / Holdoff" tab.
3. Select the "Holdoff mode".
4. Enter the "Holdoff settings" belonging to the selected mode.

6.2.4 Setting up an A → B → R trigger sequence

The complete configuration of a complex "A → B → R" trigger sequence consists of:

- A-trigger condition
- B-trigger condition in the same way as for the A-trigger, and optional delay time between the two triggers
- Optional reset by timeout and/or R-trigger

For details on sequence settings, see Chapter 6.8, "Sequence", on page 231.

1. Open the "Menu" > "Trigger" dialog.
2. Set "Trigger on" = "Sequence".
3. Select the type of "Sequence": "A → B → R".
4. Tap "Setup A trigger".

   A dialog opens where you can configure the first event.
   See: Chapter 6.2.1, "Configuring a simple trigger", on page 195.
5. Tap "Setup B trigger" and configure the B-trigger condition.

A dialog opens where you can configure the second event.

6. Optionally, set the "After 1st event wait" that the instrument waits after an A-trigger until it recognizes B-triggers.
7. Set the "2nd event count". The last B-trigger causes the trigger.
8. You can also define a reset condition. The sequence restarts with the A-trigger if no B-trigger occurs and the reset condition is fulfilled.
   a) To specify a reset by timeout, enable "Reset on timeout", and enter the "Rest timeout".
   b) To specify a reset trigger type condition, enable "Reset event" and configure the reset trigger type.
      The trigger types and settings depend on the A and B trigger settings. The instrument provides only possible, reasonable combinations.

6.2.5 Qualifying the trigger

Qualification considers the states of digital signals on other input channels and their logical combination as an additional trigger condition. For example, an edge trigger is configured for channel 1, and the instrument triggers only if the signal on channel 2 is high.

For details on qualification settings and restrictions, see Chapter 6.7, "Qualification", on page 228.

1. On the "Menu", select "Trigger" > "Qualify" tab.
2. Set the channel(s) with the input signal to be used as qualifying signal(s) to "On".
   The channel used as trigger source cannot be used for qualification.
3. Check and set the threshold levels for all used channels, that is, the thresholds for digitization of analog signals.
4. Set the signal state for each channel: high or low.
5. If more than one channel is selected, set the logical combination of the channel states.
6. If not yet enabled, tap "Qualify" to enable the qualification.

6.3 Trigger types

The setup of the trigger type is the most important part of the trigger definition. It determines the method to identify specific signal phenomena. Almost all trigger types are available for all conditions in a trigger sequence, that is, you can combine different types in the sequence. The instrument checks the trigger settings for compatibility and feasibility, and disables settings that do not fit the previous settings in the sequence.

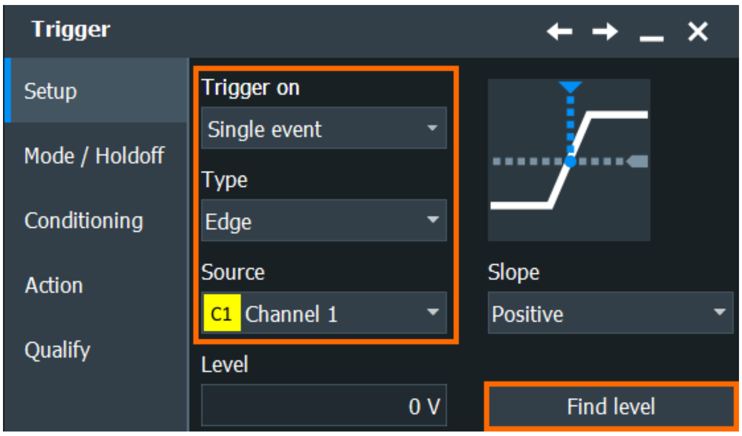
The settings in the tab are:

6.3.1 Basic trigger settings

"Setup"Access: "Menu" > "Trigger" > "Setup" tab

The basic trigger settings are the trigger source and the trigger type, including the trigger level. These settings are specific for each condition in a trigger sequence. To set the trigger level automatically, use "Find level".

Depending on the trigger type, additional settings are available. They are explained in the trigger-type specific sections.



Trigger on
Selects, if you want to trigger on a single event, or on a series of events.

Remote command:
`TRIGger<m>:EVENt` on page 1393

Source
Selects the source of the trigger signal for the current trigger condition. The trigger source works even if it is not displayed in a diagram. It must be synchronized to the signal to be displayed and analyzed.

The trigger source can be:
- Channel 1...4: An analog input channel
- Extern: External analog signal connected to the external trigger input
  The external trigger source is supported for the "Single event" sequence. It is not available if a longer trigger sequence is selected, or if qualification is enabled.
- Serial bus, D0...D15, Logic, Parallel bus 1...4:
  If options with trigger functionality are installed, the variety of trigger sources is enhanced with specific trigger sources.

Available sources depend on the trigger sequence setting. If "Single event" is selected, all inputs (analog input channels, serial and parallel buses, digital channels) can be used as trigger source. If any other trigger sequence is selected, only channel inputs Ch1...4 can be set as trigger source, and all other input sources are disabled.

See also: Chapter 6.8, "Sequence", on page 231

Remote command:
`TRIGger<m>:SOURce[:SELect]` on page 1391

Type
Selects the trigger type specific for each condition in a trigger sequence. The current trigger type is shown on the button.

The following trigger types are available:
- Edge, see page 200
- Glitch, see page 201
- Width, see page 202
- Runt, see page 203
- Window, see page 205
- Timeout, see page 206
- Interval, see page 207
- Slew rate, see page 208
- Setup & Hold trigger, see page 210
- State, see page 212
- Pattern, see page 213
- Serial pattern, see page 215
- TV/Video trigger, see page 218
- NFC trigger, see page 1063 (requires option R&S RTO-K11)
- CDR trigger, see page 1115 (requires option R&S RTO-K13)

Restrictions:
- If the external trigger input is used as trigger source, the analog edge trigger is the only available trigger type.
- For the R-trigger (reset), the trigger types and settings are restricted dependent on the A and B trigger settings. The instrument provides only possible, reasonable combinations.

Remote command:
`TRIGger<m>:TYPE` on page 1392

Find level

Sets the trigger level automatically to *0.5 * (MaxPeak – MinPeak)*. The function is not available for an external trigger source and the TV trigger.

Remote command:
`TRIGger<m>:FINDlevel` on page 1394

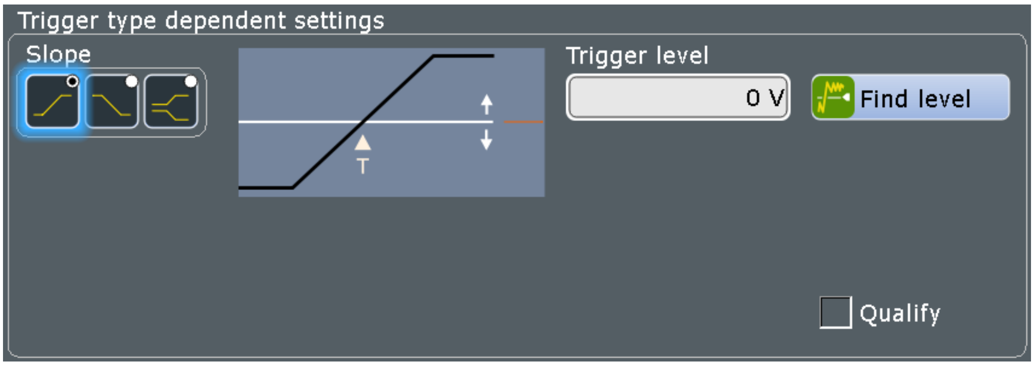
6.3.2 Edge

Access: "Menu">"Trigger" > "Setup" tab > "Type = Edge"

The edge trigger is the most common trigger type. It is well known from analog oscilloscopes; and you can use it for analog and digital signals.

The trigger condition is fulfilled when the signal from the trigger source passes the specified threshold voltage in the specified direction (slope).

If the trigger source is a channel signal, the edge trigger uses the digitized trigger signal. This signal can be qualified and filtered with the DSP filter. If the trigger source is the external trigger input, the coupling and filter for this signal is set directly in the trigger setup.



Slope

Sets the edge type for the trigger condition.

"Positive" Selects the rising edge, that is a positive voltage change.

"Negative" Selects the falling edge, that is a negative voltage change.

"Both" Selects the rising as well as the falling edge. This option is not available if the trigger source is the external trigger input.

Remote command:
`TRIGger<m>:EDGE:SLOPe` on page 1395
`TRIGger<m>:ANEDge:SLOPe` on page 1397
`TRIGger<m>:SLEW:SLOPe` on page 1408

Level

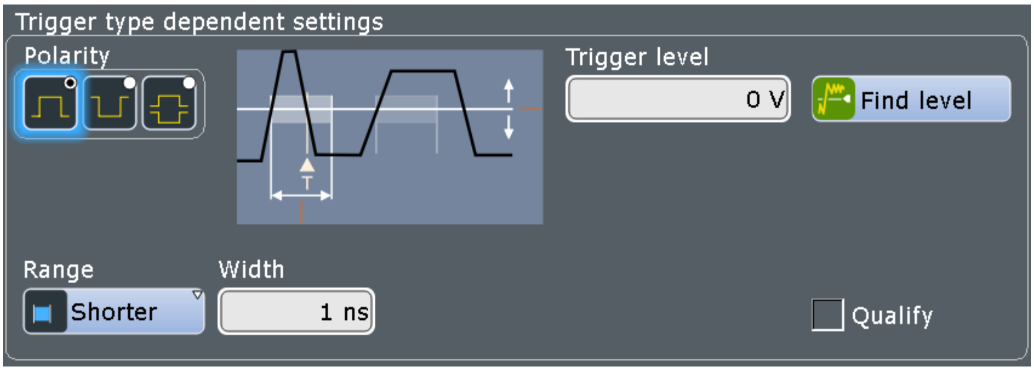
Sets the voltage level for the trigger condition. You can also drag the trigger level marker on the display (TA or TB on the right edge of the display).

Remote command:
`TRIGger<m>:LEVel<n>[:VALue]` on page 1393

6.3.3 Glitch

Access: "Menu">"Trigger" > "Setup" tab > "Type = Glitch"

The glitch trigger detects pulses shorter or longer than a specified time. It identifies deviation from the nominal data rate and helps to analyze causes of even rare glitches and their effects on other signals.



Polarity
Indicates the polarity of a pulse, that is the direction of the first pulse slope.

"Positive"	Selects positive going pulses.
"Negative"	Selects negative going pulses.
"Either"	Selects both positive and negative going pulses.

Remote command:
`TRIGger<m>:GLITch:POLarity` on page 1398
`TRIGger<m>:RUNT:POLarity` on page 1400

Range
Selects which glitches are identified: shorter or longer than the specified "Width".

Remote command:
`TRIGger<m>:GLITch:RANGe` on page 1398

Width
Sets the length of a glitch. The instrument triggers on pulses shorter or longer than this value. The minimum width is 100 ps.

You need to know the expected pulse widths of the circuit to set the glitch width correctly.

Remote command:
`TRIGger<m>:GLITch:WIDTh` on page 1398

Trigger level
Sets the voltage level for the trigger condition. You can also drag the trigger level marker on the display (TA or TB on the right edge of the display). The range of the trigger level is limited in a way so that always a hysteresis for stable trigger conditions is available.

Remote command:
TRIGger<m>:LEVel<n>[:VALue] on page 1393

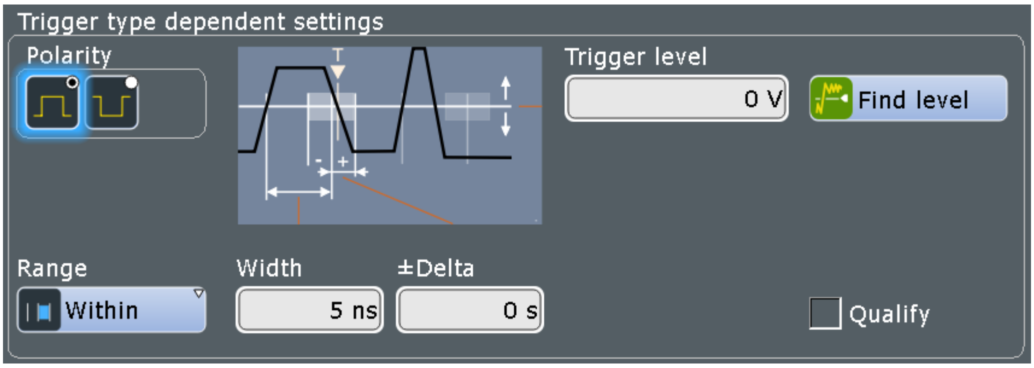
6.3.4 Width

Access: "Menu">"Trigger" > "Setup" tab > "Type = Width"

The width trigger compares the pulse width (duration of a pulse) with a given time limit. It detects pulses with an exact pulse width, pulses shorter or longer than a given time, and pulses inside or outside the allowable time range. The pulse width is measured at the trigger level.

Using the width trigger, you can define the pulse width more precisely than with the glitch trigger. However, using the range settings "Shorter" and "Longer", you can also trigger on glitches.

The width trigger can only analyze **either** positive **or** negative polarity, but searching for a width is also possible for both polarities at the same time ("Either").



Polarity
Indicates the polarity of a pulse, that is the direction of the first pulse slope.

"Positive" Triggers on positive going pulses.

"Negative" Triggers on negative going pulses.

Remote command:
TRIGger<m>:WIDTh:POLarity on page 1399

Range
Selects how the range of a pulse width is defined:

"Within" Triggers on pulses inside a given range. The range of the pulse width is defined by "±Delta" related to "Width".

"Outside" Triggers on pulses outside a given range. The range definition is the same as for "Within" range.

"Shorter" Triggers on pulses shorter than the given "Width".

"Longer" Triggers on pulses longer than the given "Width".

Remote command:
TRIGger<m>:WIDTh:RANGe on page 1399

Width
For the ranges "Within" and "Outside", the width defines the center of a range which is defined by the limits ±Delta.

For the ranges "Shorter" and "Longer", the width defines the maximum and minimum pulse width, respectively.

Remote command:
`TRIGger<m>:WIDTh:WIDTh` on page 1400

±Delta
Defines a range around the given width value.

The combination "Range" = Within and "±Delta" = 0 triggers on pulses with a pulse width that equals "Width".

The combination "Range" = Outside and "±Delta" = 0 means to trigger on pulse widths ≠ "Width".

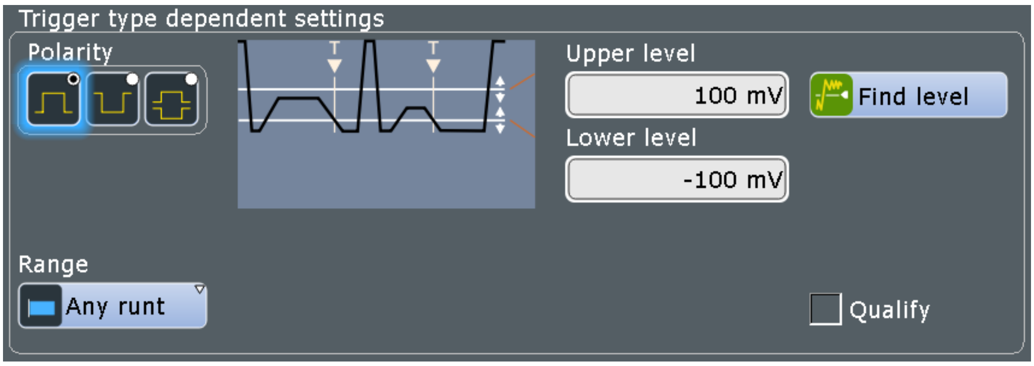
Trigger level
Sets the voltage level for the trigger condition. You can also drag the trigger level marker on the display (TA or TB on the right edge of the display). The range of the trigger level is limited in a way so that always a hysteresis for stable trigger conditions is available.

Remote command:
`TRIGger<m>:LEVel<n>[:VALue]` on page 1393

6.3.5 Runt

Access: "Menu">"Trigger" > "Setup" tab > "Type = Runt"

A runt is a pulse lower than normal in amplitude. The amplitude crosses the first threshold twice in succession without crossing the second one. In addition to the threshold amplitudes, you can define a time limit for the runt in the same way as for width triggers. For example, this trigger can detect logic, digital, and analog signals remaining below a specified threshold amplitude because I/O ports are in undefined state.



Polarity
Indicates the polarity of a pulse, that is the direction of the first pulse slope.

"Positive" Selects positive going pulses.

"Negative" Selects negative going pulses.

"Either" Selects both positive and negative going pulses.

Remote command:
TRIGger<m>:GLITch:POLarity on page 1398
TRIGger<m>:RUNT:POLarity on page 1400

Upper level
Sets the upper voltage threshold.

Remote command:
TRIGger<m>:LEVel<n>:RUNT:UPPer on page 1401

Lower level
Sets the lower voltage threshold.

Remote command:
TRIGger<m>:LEVel<n>:RUNT:LOWer on page 1401

Range
Selects how the time limit of the runt pulse is defined:

"Any runt" Triggers on all runts fulfilling the level condition, without time limitation.

"Longer" Triggers on runts longer than the given "Runt width".

"Shorter" Triggers on runts shorter than the given "Runt width".

"Within" Triggers if the runt length is inside a given time range. The range is defined by "Runt width" and "±Delta".

"Outside" Triggers if the runt length is outside a given time range. The range definition is the same as for "Within" range.

Remote command:
TRIGger<m>:RUNT:RANGe on page 1401

Runt width
For the ranges "Shorter" and "Longer", the runt width defines the maximum and minimum pulse width, respectively.

For the ranges "Within" and "Outside", the runt width defines the center of a range which is defined by "±Delta".

Remote command:
TRIGger<m>:RUNT:WIDTh on page 1402

±Delta
Defines a range around the given runt width.

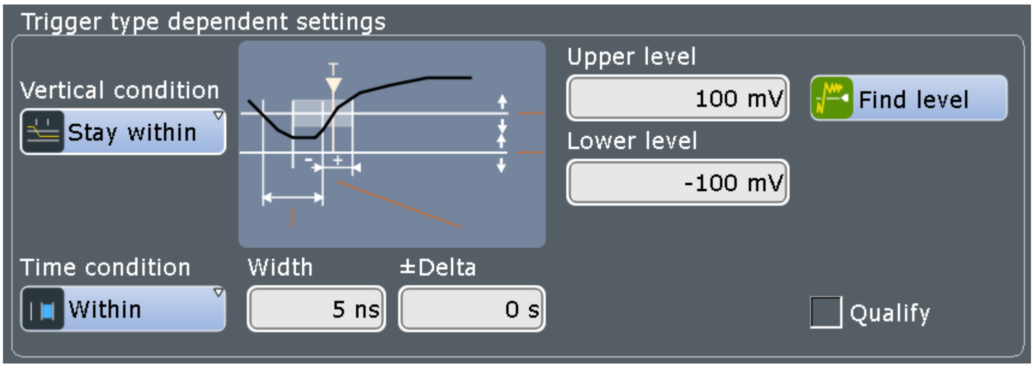
Remote command:
TRIGger<m>:RUNT:DELTa on page 1402

6.3.6 Window

Access: "Menu">"Trigger" > "Setup" tab > "Type = Window"

The window trigger checks the signal run in relation to a "window". The window is formed by the upper and lower voltage levels. The trigger condition is fulfilled, if the waveform enters or leaves the window, or if the waveform stays inside or outside for a time longer or shorter than specified.

With the window trigger, you can display longer transient effects.



Vertical condition
Selects how the signal run is compared with the window:

"Enter" Triggers when the signal crosses the upper or lower level and thus enters the window made up of these two levels.

"Exit" Triggers when the signal leaves the window.

"Stay within" Triggers if the signal stays between the upper and lower level for a specified time. The time is defined in various ways by the Time condition.

"Stay outside" Triggers if the signal stays above the upper level or below the lower level for a specified time. The time is also defined by the "Time condition".

Remote command:
`TRIGger<m>:WINDow:RANGe` on page 1403

Upper level
Sets the upper voltage limit for the window.

Remote command:
`TRIGger<m>:LEVel<n>:WINDow:UPPer` on page 1403

Lower level
Sets the lower voltage limit for the window.

Remote command:
`TRIGger<m>:LEVel<n>:WINDow:LOWer` on page 1403

Time condition
Selects how the time limit of the window is defined. Time conditioning is available for the vertical conditions "Stay within" and "Stay outside".

"Within" Triggers if the signal stays inside or outside the vertical window limits at least for the time *Width* - *Delta* and for *Width* + *Delta* at the most.

"Outside" "Outside" is the opposite definition of "Within". The instrument triggers if the signal stays inside or outside the vertical window limits for a time shorter than *Width* - *Delta* or longer than *Width* + *Delta*.

"Shorter" Triggers if the signal crosses vertical limits before the specified "Width" time is reached.

"Longer" Triggers if the signal crosses vertical limits after the specified "Width" time is reached.

Remote command:
TRIGger<m>:WINDow:TIME on page 1404

Width
For the ranges "Within" and "Outside", the width defines the center of a time range which is defined by the limits "±Delta".

For the ranges "Shorter" and "Longer", it defines the maximum and minimum time lapse, respectively.

Remote command:
TRIGger<m>:WINDow:WIDTh on page 1404

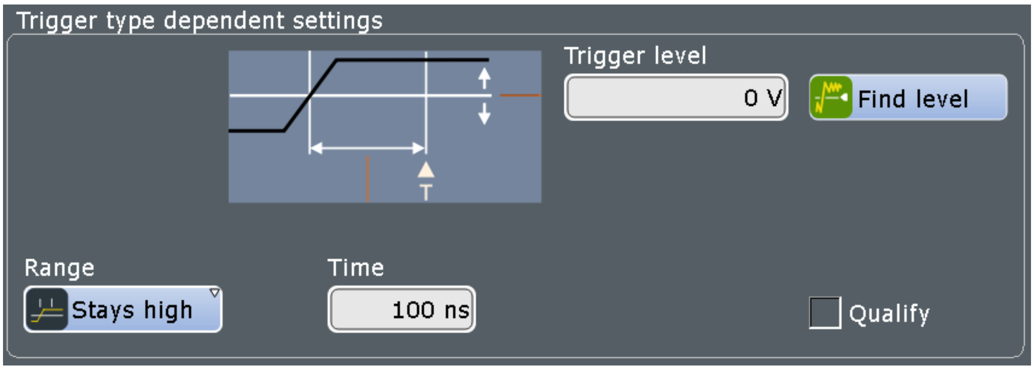
±Delta
Defines a range around the "Width" value.

Remote command:
TRIGger<m>:WINDow:DELTa on page 1405

6.3.7 Timeout

Access: "Menu">"Trigger" > "Setup" tab > "Type = Timeout"

The timeout trigger checks if the signal stays above or below the threshold voltage for a specified time lapse. In other words, the trigger occurs if the trigger source does not have the expected transition within the specified time.



Trigger level
Sets the voltage level for the trigger condition. You can also drag the trigger level marker on the display (TA or TB on the right edge of the display). The range of the trigger level is limited in a way so that always a hysteresis for stable trigger conditions is available.

Remote command:
`TRIGger<m>:LEVel<n>[:VALue]` on page 1393

Range
Selects the relation of the signal level to the trigger level:

"Stays high" The signal level stays above the trigger level.

"Stays low" The signal level stays below the trigger level.

"High or low" The signal level stays above or below the trigger level.

Remote command:
`TRIGger<m>:TIMeout:RANGe` on page 1405

Time
Defines the time limit for the timeout at which the instrument triggers.

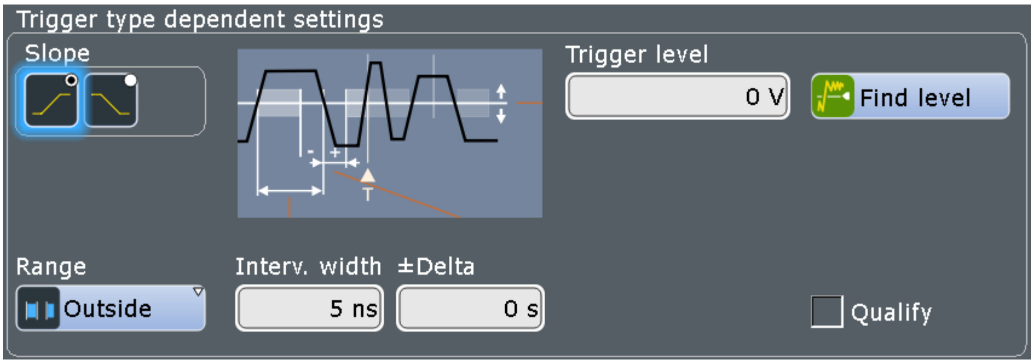
Remote command:
`TRIGger<m>:TIMeout:TIME` on page 1406

6.3.8 Interval

Access: "Menu">"Trigger" > "Setup" tab > "Type = Interval"

The interval trigger analyzes the time between two pulses.

The interval trigger can analyze either rising or falling edges, but searching for an interval is also possible for both edges at the same time ("Either").



Slope
Sets the edge for the trigger. You can analyze the interval between positive edges or between negative edges.

Remote command:
`TRIGger<m>:INTerval:SLOPe` on page 1406

Trigger level
Sets the voltage level for the trigger condition. You can also drag the trigger level marker on the display (TA or TB on the right edge of the display). The range of the trigger level is limited in a way so that always a hysteresis for stable trigger conditions is available.

Remote command:
`TRIGger<m>:LEVel<n>[:VALue]` on page 1393

Range
Selects how the range of an interval is defined:

"Within"	Triggers on pulse intervals inside a given range. The range is defined by "Interv. width" and "±Delta".
"Outside"	Triggers on intervals outside a given range. The range definition is the same as for "Within" range.
"Shorter"	Triggers on intervals shorter than the given "Interv. width".
"Longer"	Triggers on intervals longer than the given "Interv. width".

Remote command:
`TRIGger<m>:INTerval:RANGe` on page 1406

Interv. width
Defines the time between two pulses.

Remote command:
`TRIGger<m>:INTerval:WIDTh` on page 1407

±Delta
Defines a range around the "Interval width" value.

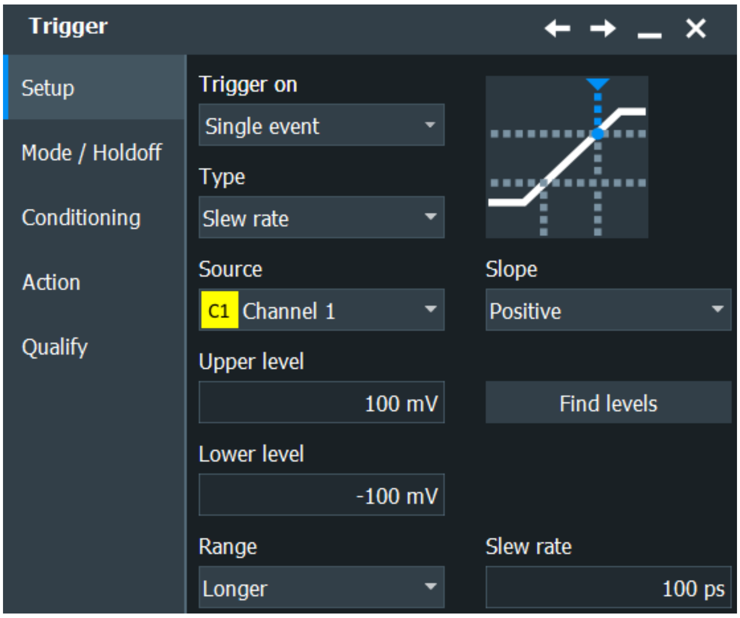
Remote command:
`TRIGger<m>:INTerval:DELTa` on page 1407

6.3.9 Slew rate

Access: "Menu">"Trigger" > "Setup" tab > "Type = Slew rate"

The slew rate trigger is also known as transition trigger. It triggers if the transition time from the lower to higher voltage level (or vice versa) is shorter or longer as defined, or outside or inside a specified time range.

The slew rate trigger finds slew rates faster than expected or permissible to avoid overshooting and other interfering effects. It also detects slow edges violating the timing in pulse series.



Slope

Sets the edge type for the trigger condition.

"Positive" Selects the rising edge, that is a positive voltage change.

"Negative" Selects the falling edge, that is a negative voltage change.

"Both" Selects the rising as well as the falling edge. This option is not available if the trigger source is the external trigger input.

Remote command:
`TRIGger<m>:EDGE:SLOPe` on page 1395
`TRIGger<m>:ANEDge:SLOPe` on page 1397
`TRIGger<m>:SLEW:SLOPe` on page 1408

Upper level

Sets the upper voltage threshold. When the signal crosses this level, the slew rate measurement starts or stops depending on the selected slope.

Remote command:
`TRIGger<m>:LEVel<n>:SLEW:UPPer` on page 1408

Lower level

Sets the lower voltage threshold. When the signal crosses this level, the slew rate measurement starts or stops depending on the selected slope.

Remote command:
`TRIGger<m>:LEVel<n>:SLEW:LOWer` on page 1408

Range

Selects how the time limit for the slew rate is defined. The time measurement starts when the signal crosses the first trigger level - the upper or lower level depending on the selected slope. The measurement stops when the signal crosses the second level.

"Within" Triggers on slew rates inside a given time range. The range is defined by "Slew rate" and "±Delta".

"Outside" Triggers on slew rates outside a given time range. The range definition is the same as for "Within" range.

"Shorter" Triggers on slew rates shorter than the given "Slew rate" limit.

"Longer" Triggers on slew rates longer than the given "Slew rate" limit.

Remote command:
`TRIGger<m>:SLEW:RANGe` on page 1409

Slew rate
For the ranges "Within" and "Outside", the slew rate defines the center of a range which is defined by the limits "±Delta".

For the ranges "Shorter" and "Longer", the slew rate defines the maximum and minimum slew rate limits, respectively.

Remote command:
`TRIGger<m>:SLEW:RATE` on page 1409

±Delta
Defines a time range around the given slew rate.

Remote command:
`TRIGger<m>:SLEW:DELTa` on page 1410

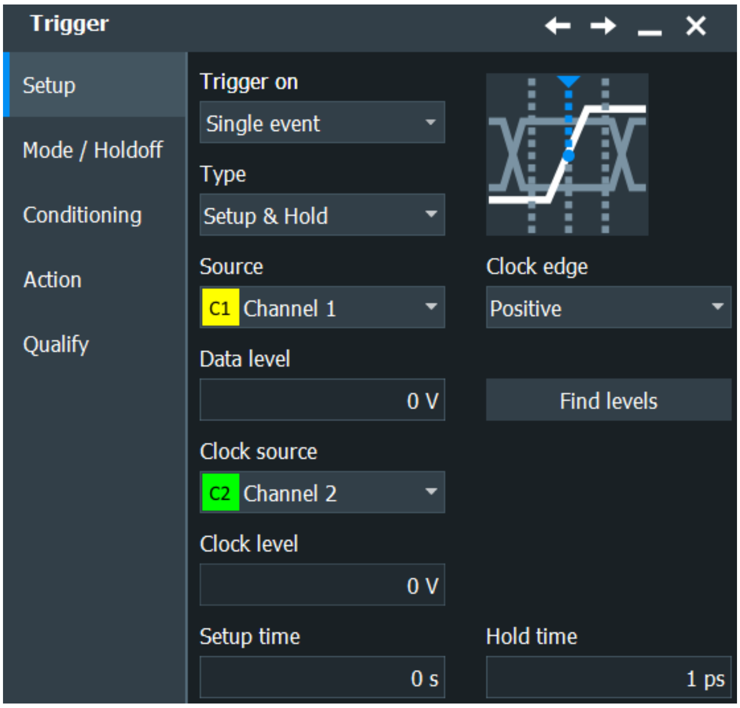
6.3.10 Setup & Hold trigger

Access: "Menu">"Trigger" > "Setup" tab > "Type = Setup & Hold"

With the Setup & Hold trigger you can analyze the relative timing between two signals: a data signal and the synchronous clock signal. Many systems require, that the data signal must be steady for some time before and after the clock edge, for example, the data transmission on parallel interfaces. With this trigger type, you can also test the time correlation of sideband and in-band signals.

In firmware versions < 5.00, the Setup & Hold was called Data2Clock trigger.

The trigger occurs if the data signal crosses the data level during the setup and hold time. The reference point for the time measurement is defined by clock level and clock edge.



Clock source
Selects the input channel of the clock signal.

Remote command:
`TRIGger<m>:SETHold:CSOurce[:VALue]` on page 1411
`TRIGger<m>:SPATtern:CSOurce[:VALue]` on page 1415

Clock edge
Sets the edge of the clock signal to define the time reference point for the setup and hold time:

"Positive"	Rising edge, a positive voltage change.
"Negative"	Falling edge, a negative voltage change.
"Both"	Both the rising and the falling edge.

Remote command:
`TRIGger<m>:SETHold:CSOurce:EDGE` on page 1410

Clock level
Sets the voltage level for the clock signal. Both "Clock level" and "Clock edge" define the starting point for calculation of the setup and hold time.

Remote command:
`TRIGger<m>:SETHold:CSOurce:LEVel` on page 1410

Data level
Sets the voltage level for the data signal. At this level, the setup and hold time is measured.

Remote command:
`TRIGger<m>:LEVel<n>[:VALue]` on page 1393

Couple levels (Trigger level and hysteresis coupling)
Sets the trigger levels and hysteresis values for all channels to the values of the currently selected trigger source. The function affects only the levels defined for the selected condition. The hysteresis of the external trigger input is an independent value, and it is not affected by level coupling.

In trigger sequences, another coupling of trigger levels is possible: "Couple sequence thresholds" on page 234.

Remote command:
`TRIGger<m>:SCOupling` on page 1412

Setup time
Sets the minimum time **before** the clock edge while the data signal must stay steady above or below the data level.

The setup time can be negative. In this case, the hold time is always positive. If you set a negative setup time, the hold time is adjusted by the instrument.

Remote command:
`TRIGger<m>:SETHold:STIMe` on page 1411

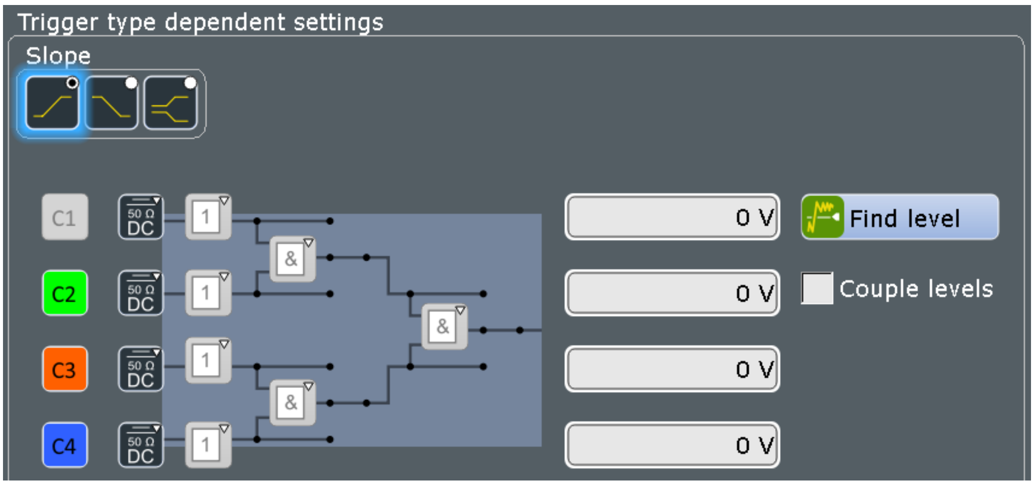
Hold time
Sets the minimum time **after** the clock edge while the data signal must stay steady above or below the data level.

The hold time can be negative. In this case, the setup time is always positive. If you set a negative hold time, the setup time is adjusted by the instrument.

Remote command:
`TRIGger<m>:SETHold:HTIMe` on page 1411

6.3.11 State

The state trigger is a qualified edge trigger. It combines the edge trigger settings with trigger qualification.



The individual settings are:

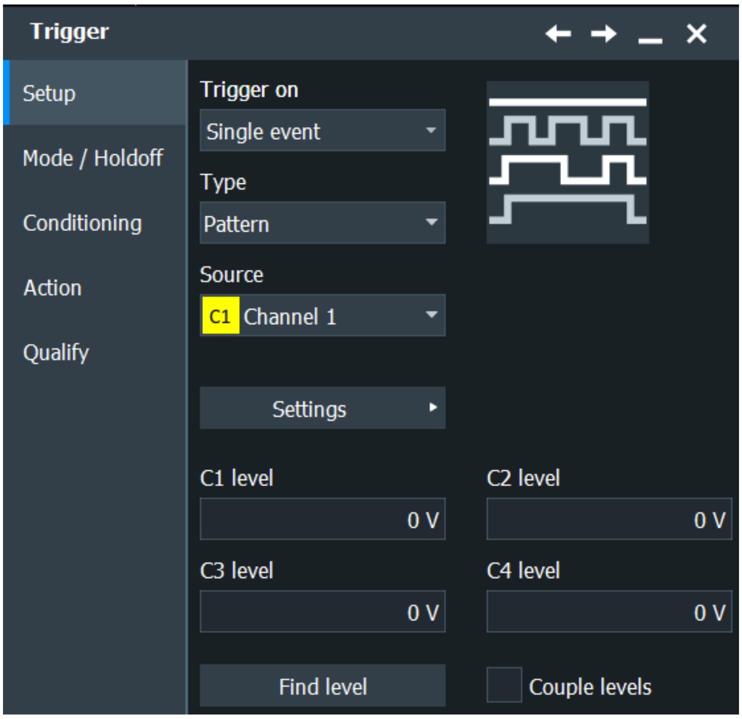
- "Slope" on page 200
- "Pattern" on page 213
- "Trigger Levels" on page 214
- "Find level" on page 200
- "Couple levels (Trigger level and hysteresis coupling)" on page 212

6.3.12 Pattern

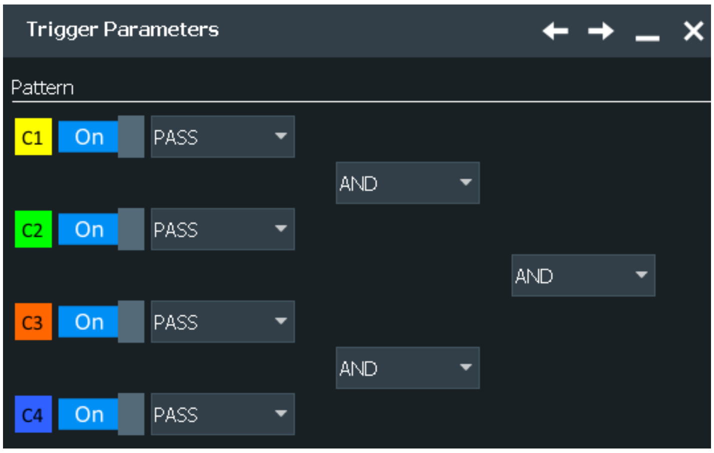
Access: "Menu">"Trigger" > "Setup" tab > "Type = Pattern"

The pattern trigger is a logic trigger. It provides logical combination of the input channels and supports you in verifying the operation of digital logic. In addition to the pattern and the trigger levels, you can define a timing condition.

The setup of the pattern trigger is similar to trigger qualification. The complete settings for the pattern trigger are provided in the "Qualification" tab.



Pattern
The pattern contains the channel selection, and the logical operations structure of hardware-based Boolean logic.



"Channel"
Select the channels to be considered. For qualification, you can select all channel signals except for the trigger source. In pattern trigger setup, the trigger source channel is selected by default, and you can select all other channel signals.

"Boolean operator"
Defines the logical operation on the digital signal resulting from the comparison with the trigger level.

- "PASS": leaves the input value unchanged
- "NOT": inverts the input value

"Logical operator"
Defines the logic combination of two sources. The sources are channel 1/2 and channel 3/4 on the first step, and in the second step the logical combination resulting from the first step.

- "AND": logical AND, conjunctive combination
- "NAND": logical NOT AND
- "OR": logical OR, disjunctive combination
- "NOR": logical NOT OR

Remote command:
TRIGger<m>:QUALify<n>:A:LOGic on page 1424
TRIGger<m>:QUALify<n>:A[:ENABle] on page 1423
TRIGger<m>:QUALify<n>:AB:LOGic on page 1425
TRIGger<m>:QUALify<n>:ABCD:LOGic on page 1425
TRIGger<m>:QUALify<n>:B:LOGic on page 1424
TRIGger<m>:QUALify<n>:B[:ENABle] on page 1423
TRIGger<m>:QUALify<n>:C:LOGic on page 1424
TRIGger<m>:QUALify<n>:C[:ENABle] on page 1423
TRIGger<m>:QUALify<n>:CD:LOGic on page 1425
TRIGger<m>:QUALify<n>:D:LOGic on page 1424
TRIGger<m>:QUALify<n>:D[:ENABle] on page 1423

Trigger Levels
Define the trigger level for each input channel. For state and pattern trigger, the trigger level is a decision threshold: If the signal value is higher than the trigger level, the signal state is high (1 or true for the Boolean logic). Otherwise, the signal state is considered low (0 or false) if the signal value is below the trigger level.

These trigger levels are also used in qualification setup.

You can set the trigger levels for all channels to the same value, see "Couple levels (Trigger level and hysteresis coupling)" on page 212.

Additional settings: Timing
"Timing" adds time limitation to the pattern condition.

You find this setting in the "Qualification" tab.

"Off"	No time limitation. The trigger occurs if the pattern condition is fulfilled.
"Timeout"	Defines how long the result of the pattern condition must be true or false.
"Width"	Defines a time range for keeping up the true result of the pattern condition. The range is defined in the same way as for width and interval triggers, see "Range" on page 202.

Remote command:
`TRIGger<m>:PATTern:MODE` on page 1413
`TRIGger<m>:PATTern:TIMeout:MODE` on page 1413
`TRIGger<m>:PATTern:TIMeout[:TIME]` on page 1414
`TRIGger<m>:PATTern:WIDTh:DELTa` on page 1415
`TRIGger<m>:PATTern:WIDTh:RANGe` on page 1414
`TRIGger<m>:PATTern:WIDTh[:WIDTh]` on page 1415

6.3.13 Serial pattern

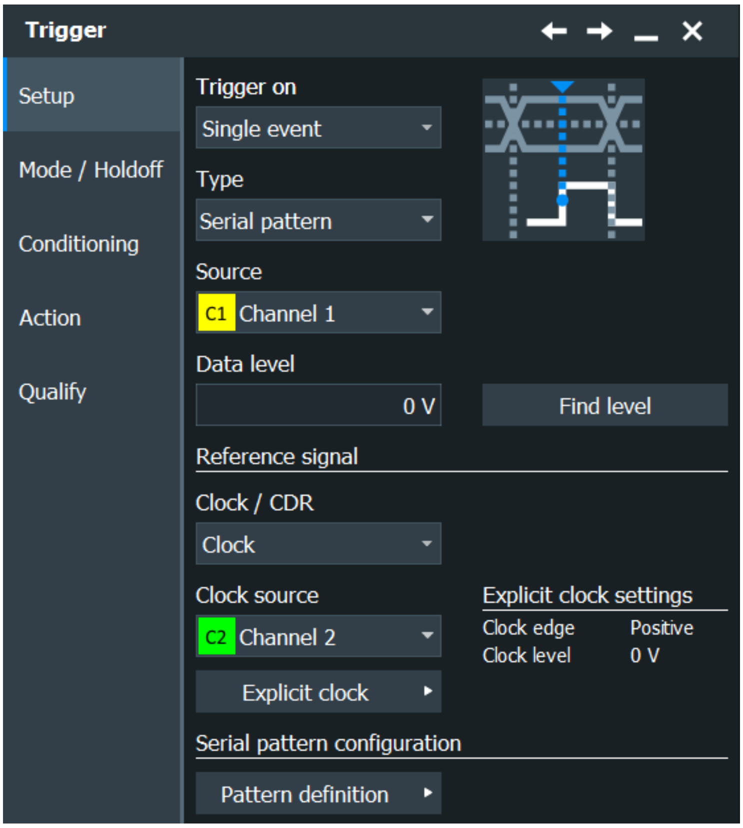
Access: "Menu" > "Trigger" > "Setup" tab > "Type = Serial Pattern"

The serial pattern is used to trigger on signals with serial data patterns in relation to a clock signal - for example, on bus signals like the I²C bus.

The instrument expects the bits coming in LSB first order. A triggered waveform in the diagram shows the LSB on the left and the MSB on the right side.

For convenient and comprehensive triggering on specific serial data, options for serial protocol analysis are provided.

If option R&S RTO--K13 is installed, you can use the recovered clock signal for the serial pattern trigger instead of a real clock signal, see Chapter 18.3.2.3, "Serial pattern trigger using CDR", on page 1116.



Data level

Sets the voltage level for the data signal.

If the signal value is higher than the data level, the state is 1. Below the level, the signal state is 0.

Remote command:
`TRIGger<m>:LEVel<n>[:VALue]` on page 1393

Clock / CDR

Selects if an explicit clock or a CDR signal is used as a reference signal for the serial pattern trigger.

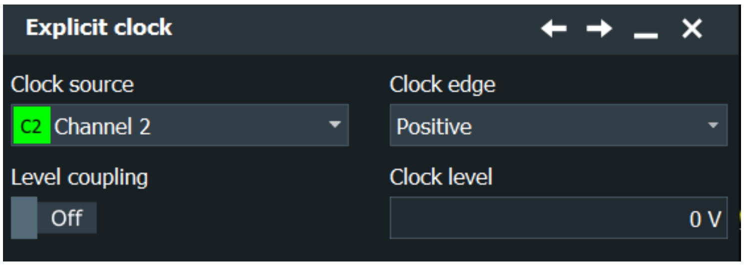
"CDR" Uses clock data recovery. See Chapter 18.3.2.3, "Serial pattern trigger using CDR", on page 1116.

"Clock" Uses an explicit clock signal, see Explicit clock.

Remote command:
`TRIGger<m>:SPATtern:REFSource` on page 1417

Explicit clock

Opens a dialog to define the settings of the explicit clock settings.



Clock source ← Explicit clock
Selects the input channel of the clock signal.

Remote command:
`TRIGger<m>:SETHold:CSOurce[:VALue]` on page 1411
`TRIGger<m>:SPATtern:CSOurce[:VALue]` on page 1415

Clock edge ← Explicit clock
Together with the clock level, the clock edge sets the point in time when the state of the data signal is checked:

"Positive"	Rising edge, a positive voltage change.
"Negative"	Falling edge, a negative voltage change.
"Both"	Both the rising and the falling edge.

Remote command:
`TRIGger<m>:SPATtern:CSOurce:EDGE` on page 1416

Clock level ← Explicit clock
Sets the voltage level for the clock signal.

Remote command:
`TRIGger<m>:SPATtern:CSOurce:LEVel` on page 1416

Couple levels (Trigger level and hysteresis coupling) ← Explicit clock
Sets the trigger levels and hysteresis values for all channels to the values of the currently selected trigger source. The function affects only the levels defined for the selected condition. The hysteresis of the external trigger input is an independent value, and it is not affected by level coupling.

In trigger sequences, another coupling of trigger levels is possible: "Couple sequence thresholds" on page 234.

Remote command:
`TRIGger<m>:SCOupling` on page 1412

Pattern
The pattern contains the bits of the serial data to be found in the data stream. The maximum length of the pattern is 128 bit. Touch and hold the "Pattern" field to open the "Bit Pattern Editor" where you can enter the pattern in various formats.

See also: Chapter 13.1.7, "Bit pattern editor", on page 488.

In binary format, an *X* indicates that the logical level for the bit is not relevant (do not care).

Remote command:
`TRIGger<m>:SPATtern:PATTern` on page 1416

6.3.14 TV/Video trigger

Access: "Menu">"Trigger" > "Setup" tab > "Type = TV"

The TV or video trigger is used to analyze analog baseband video signals. You can trigger on baseband video signals from standard definition and high definition standards, and also on user defined signals.

The instrument triggers on the line start - the horizontal sync pulse. You can trigger on all lines, or specify a line number. You can also trigger on the field or frame start.

Also, a delay can be set: Set the "Holdoff events" in the "Mode / Holdoff" tab to the number of fields to be skipped.

For details on holdoff settings, see Chapter 6.4, "Mode / Holdoff", on page 223.

Make sure that the trigger level crosses the synchronizing pulses of the video signal, see "Trigger level" on page 220.

Once the trigger is set correctly, you can use cursor and automatic measurements to perform amplitude and timing measurements.

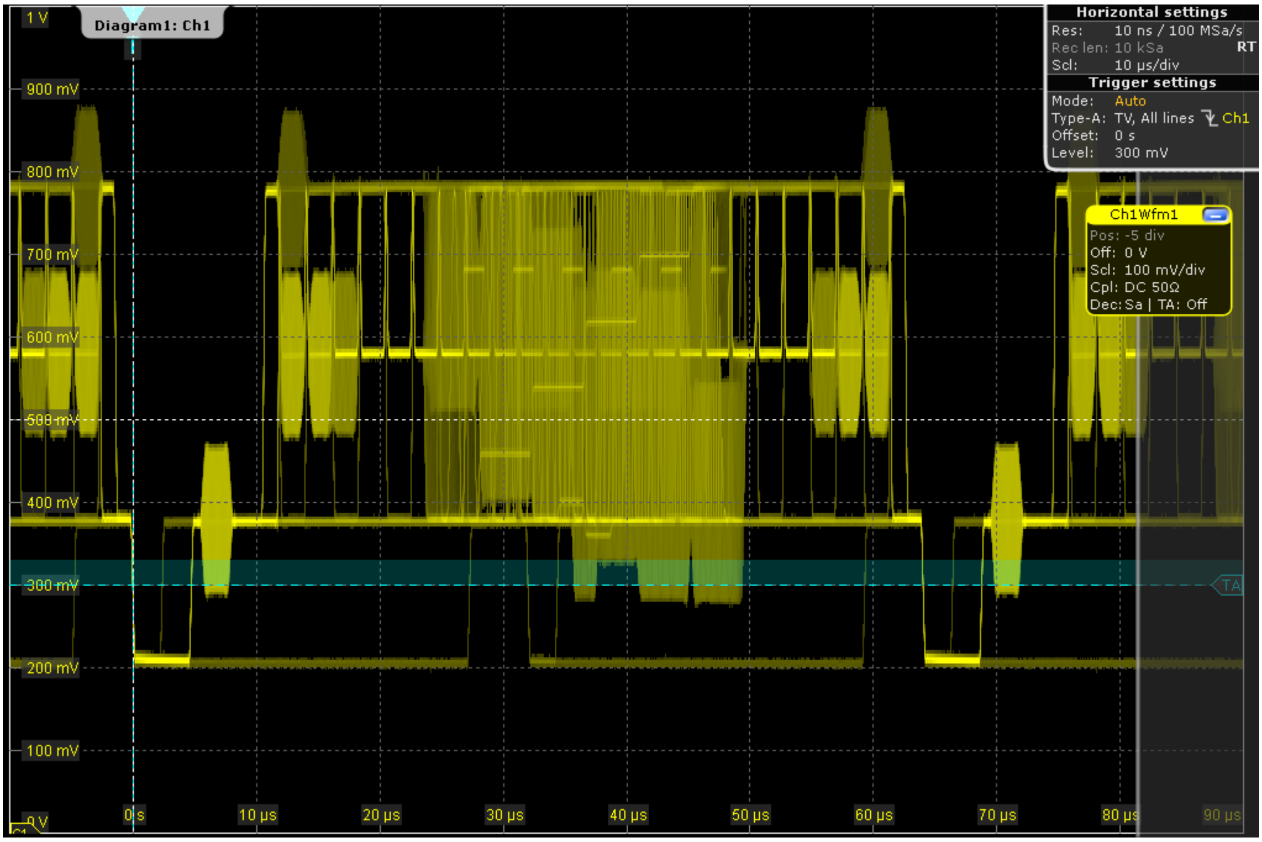


Figure 6-1: Trigger on all lines of a PAL signal with positive signal polarity, trigger level = 300 mV

Standard
Selects the TV standard or "Custom" for user-defined signals.

HDTV standards are indicated by the number of active lines, the scanning system (p for progressive scanning, i for interlaced scanning) and the frame rate. For interlaced scanning, the field rate is used instead of the frame rate. 1080p/24sF is an HDTV standard using progressive segmented frame scanning.

"Custom" can be used for signals of other video systems, for example, medical displays, video monitors, and security cameras. To trigger on these signals, you have to define the pulse type and length of the sync pulse, the scanning system and the line period.

Remote command:
`TRIGger<m>:TV:STANdard` on page 1417

Mode
Selects the lines or fields on which the instrument triggers. Available modes depend on the scanning system of the selected standard.

"All fields" Triggers on the first video line of the frame (progressive scanning) or field (interlaced scanning), for example, to find amplitude differences between the fields.

"Odd fields / Even fields"	Triggers on the first video line of the odd or even field. These modes are available for interlaced scanning (PAL, PAL-M, SECAM, NTSC, 1080i) and progressive segmented frame scanning (1080p/24sF). They can be used, for example, to analyze the components of a video signal.
"All lines"	Triggers on the line start of all video lines, for example, to find maximum video levels.
"Line number"	Triggers on a specified line. Enter the line number in "Line #".

Remote command:
TRIGger<m>:TV:MODE on page 1418

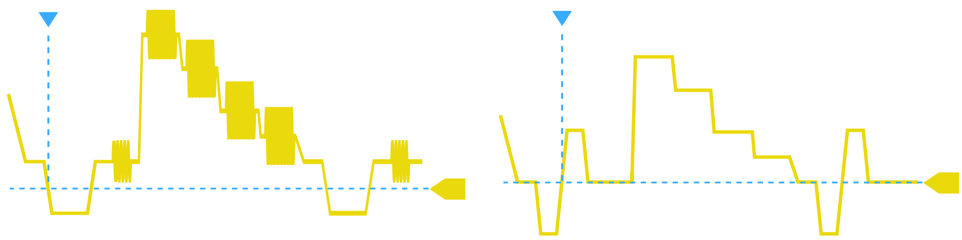
Line #
Sets the number of the line to be triggered on if "Mode" is set to "Line number". Usually the lines of the frame are counted, beginning from the frame start.

For NTSC signals, the lines are counted per field, not per frame. Therefore, you have to set the "Field" (odd or even), and the line number in the field.

Remote command:
TRIGger<m>:TV:LINE on page 1419
TRIGger<m>:TV:LFIeld on page 1420

Trigger level
Sets the trigger level as threshold for the sync pulse. Make sure that the trigger level crosses the synchronizing pulses of the video signal.

The hysteresis is set according to the settings in the "Noise Reject" tab.



Remote command:
TRIGger<m>:LEVel<n>[:VALue] on page 1393

Signal polarity
Sets the polarity of the signal. Note that the sync pulse has the opposite polarity, for example, a positive signal has a negative sync pulse.

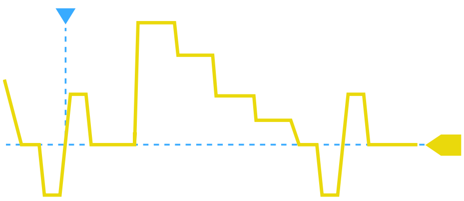


Figure 6-2: Signal with positive polarity and tri-level sync pulse

Remote command:
`TRIGger<m>:TV:POLarity` on page 1419

Pulse type

Sets the type of the sync pulse, either bi-level sync pulse (used in SDTV signals), or tri-level sync pulse (used in HDTV signals).

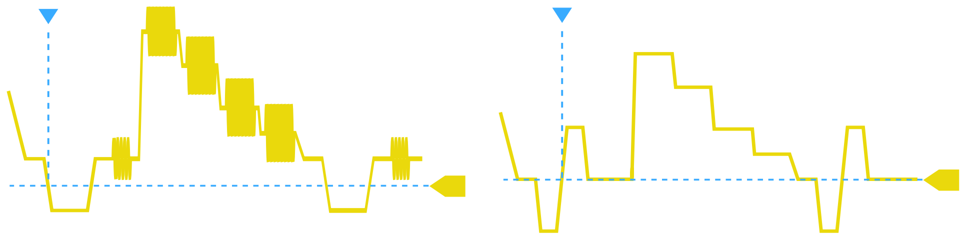


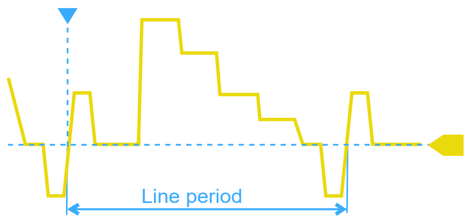
Figure 6-3: Bi-level (left) and tri-level (right) sync pulses

This setting is available for user-defined video signals if "Standard" is set to "Custom".

Remote command:
`TRIGger<m>:TV:CUSTom:STYPe` on page 1421

Line period

Sets the duration of a single video line, the time between two successive sync pulses.

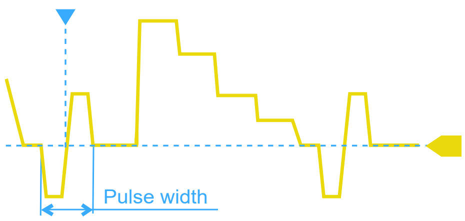


This setting is available for user-defined video signals if "Standard" is set to "Custom".

Remote command:
`TRIGger<m>:TV:CUSTom:LDURation` on page 1421

Pulse width

Sets the width of the sync pulse.



This setting is available for user-defined video signals if "Standard" is set to "Custom".

Remote command:
`TRIGger<m>:TV:CUSTom:SDURation` on page 1421

Scan

Sets the scanning system.

This setting is available for user-defined video signals if "Standard" is set to "Custom".

"Interlaced" Interlace scanning uses two fields to create a frame. One field contains all the odd lines (odd, first, or upper filed), the other contains all the even lines of the image (even, second, or lower field). First the lines of the odd filed are processed, then the lines of the even field.

"Progressive" Progressive scanning is a method to capture, transmit and display all lines of a frame in sequence.

"Segmented" Progressive segmented frame uses progressive scanning to capture the frame, and interlaced scanning for transmission and display.

Remote command:
`TRIGger<m>:TV:CUSTom:SCANmode` on page 1420

6.3.15 NFC trigger

The Near Field Communication (NFC) trigger triggers on characteristic events of NFC signals. This trigger type requires option R&S RTO--K11 "I/Q Software Interface".

For details, see Chapter 17.4, "NFC trigger", on page 1063

6.3.16 CDR trigger

The clock data recovery (CDR) trigger triggers on the edges of a clock edge stream that is recovered from a data signal using the hardware CDR. This trigger type requires option R&S RTO--K13 "CDR".

For details, see Chapter 18.3.2.2, "CDR trigger", on page 1115.

6.3.17 Triggering on serial buses

Protocol analysis including configuration, triggering, and decoding is described in Chapter 13, "Protocol analysis", on page 480.

For information on triggering on serial buses, see the "Trigger" chapter of the relevant protocol.

6.3.18 Triggering on parallel buses and digital channels

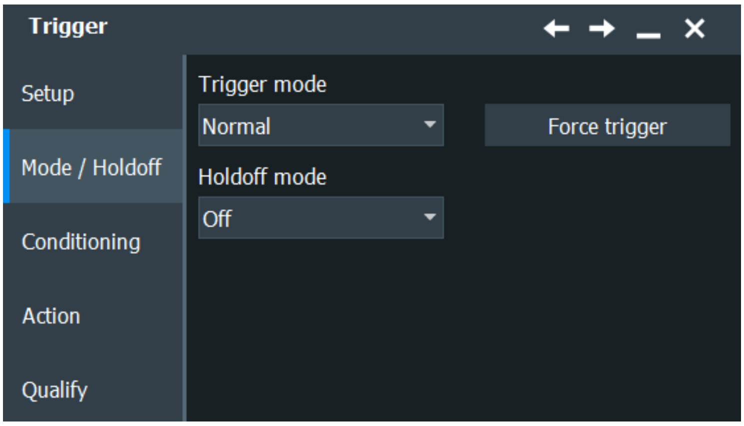
Triggering on digital signals requires the Mixed Signal Option. The option is described in Chapter 14, "Mixed signal option (MSO, R&S RTO-B1)", on page 979.

For information on triggering, see Chapter 14.3.1, "Trigger settings for digital signals and parallel buses", on page 992.

6.4 Mode / Holdoff

Access: "Trigger" menu > "Mode / Holdoff" tab

Holdoff conditions define a waiting time after the current trigger until the next trigger can be recognized.



[Run Stop]/[Single]

Front panel keys to start and stop a continuous acquisition or a defined number of acquisition cycles, respectively. The number of acquisitions is set with "Average count".

Remote command:
`RUN` on page 1326
`SINGle` on page 1326
`STOP` on page 1327

Trigger mode

Sets the trigger mode which determines the behavior of the instrument if no trigger occurs. The current setting is shown on the trigger label on top of the signal bar.

To toggle quickly between "Auto" and "Normal" mode, use the [Mode] key on the front panel (in "Trigger" section).

"Auto"	The instrument triggers repeatedly after a time interval if the trigger conditions are not fulfilled. If a real trigger occurs, it takes precedence. This mode helps to see the waveform even before the trigger conditions are set correctly. The waveform on the screen is not synchronized, and successive waveforms are not triggered at the same point of the waveform. The time interval depends on the time base settings.
"Normal"	The instrument acquires a waveform only if a trigger occurs, that is, if all trigger conditions are fulfilled. If no trigger occurs, no waveform is acquired and the last acquired waveform is displayed. If no waveform was captured before, none is displayed. When no trigger has been found for longer than one second, a message box appears that shows the time elapsed since the last trigger.

"Repetitive" The instrument starts acquisition immediately and triggers after a short time interval independent of the time base settings and faster than in "Auto" mode. Real triggers are ignored. Use this mode if the "Auto" mode is too slow.

Remote command:
`TRIGger<m>:MODE` on page 1434

Force Trigger

If the acquisition is running in normal mode and no valid trigger occurs, forcing the trigger provokes an immediate single acquisition. Thus you can confirm that a signal is available and use the waveform display to determine how to trigger on it.

If you need this function frequently, you can add the "Force Trigger" icon to the toolbar.

Remote command:
`TRIGger<m>:FORCe` on page 1435

Holdoff mode

Selects the method to define the holdoff condition.

The trigger holdoff defines when the next trigger after the current will be recognized. Thus, it affects the next trigger to occur after the current one. Holdoff helps to obtain stable triggering when the oscilloscope is triggering on undesired events.

Holdoff settings are not available if the trigger source is an external trigger input or serial bus. For the TV trigger, only the "Events" mode is useful.

Example:

You want to analyze the first pulse in a burst of several pulses. At first, you select a sufficiently slow time base to display the entire burst. Then, you set the holdoff time a little longer than the length of the burst. Now, each trigger corresponds to the first pulse in successive bursts, and you can change the time base to display the waveform in more detail.

The following methods are available:

"Time" Defines the holdoff directly as a time period. The next trigger occurs only after the "Holdoff time" has passed.

"Events" Defines the holdoff as a number of trigger events. The next trigger only occurs when this number of events is reached. The number of triggers to be skipped is defined in "Holdoff events".

"Random" Defines the holdoff as a random time limited by "Minimum time" and "Maximum time". For each acquisition cycle, the instrument selects a new random holdoff time from the specified range.
Random holdoff prevents synchronization to discover effects invisible with synchronized triggering, for example, the features of a pulse train.

"Auto" The holdoff time is calculated automatically based on the current horizontal scale.
"Auto time scaling" defines the factor the horizontal scale is multipied with.
"Auto time" shows the resulting holdoff time: *Auto time = Auto time scaling * Horizontal scale*.

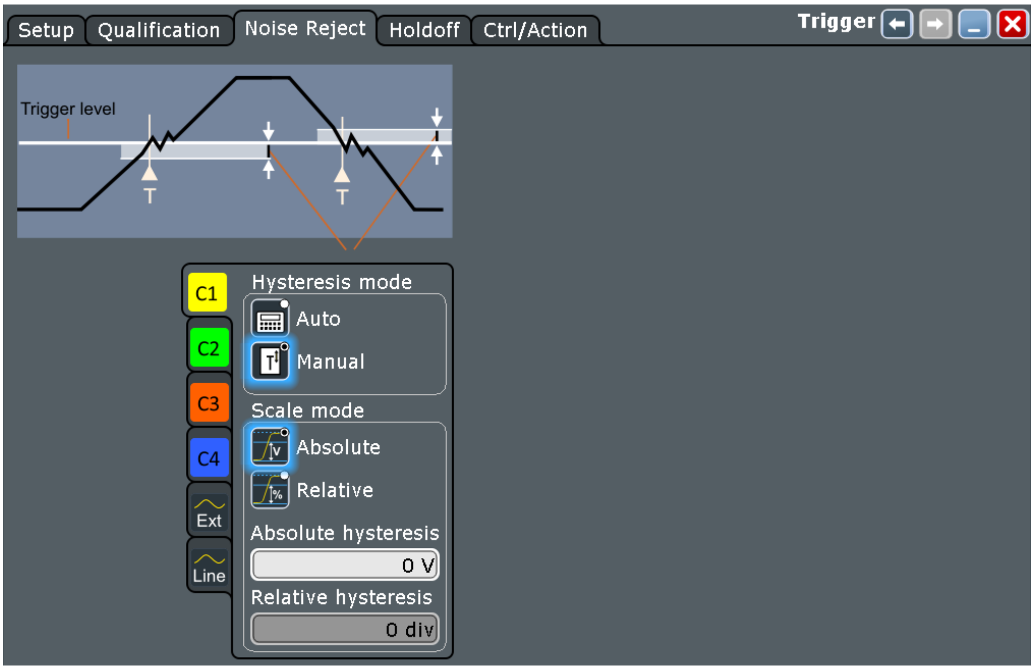
"Off" No holdoff

Remote command:
TRIGger<m>:HOLDoff:MODE on page 1425
TRIGger<m>:HOLDoff:TIME on page 1426
TRIGger<m>:HOLDoff:EVENts on page 1427
TRIGger<m>:HOLDoff:MAX on page 1428
TRIGger<m>:HOLDoff:MIN on page 1427
TRIGger<m>:HOLDoff:AUTotime? on page 1428
TRIGger<m>:HOLDoff:SCALing on page 1428

6.5 Conditioning

The rejection of noise by setting a hysteresis avoids unwanted trigger events caused by noise oscillation around the trigger level.

You can select the hysteresis mode and value for each channel separately, or couple the trigger levels and set the same hysteresis for channels. The hysteresis of the external trigger input is an independent value, and it is not affected by level coupling.



Hysteresis mode
Selects how the hysteresis is set.

"Auto" This is the recommended mode. The hysteresis is set by the instrument to reject the internal noise of the instrument.

"Manual" The hysteresis is defined directly in absolute or relative values.

Remote command:
TRIGger<m>:LEVel<n>:NOISe[:STATe] on page 1429

Size mode
Selects whether the hysteresis is defined in absolute or relative values. The setting is available only in manual hysteresis mode.

Remote command:
TRIGger<m>:LEVel<n>:NOISe:MODE on page 1430

Absolute hysteresis
Defines a range in absolute values around the trigger level. If the signal jitters inside this range and crosses the trigger level thereby, no trigger event occurs.

Remote command:
TRIGger<m>:LEVel<n>:NOISe:ABSolute on page 1430

Relative hysteresis
Defines a range in divisions around the trigger level. If the signal oscillates inside this range and crosses the trigger level thereby, no trigger event occurs.

Remote command:
TRIGger<m>:LEVel<n>:NOISe:PERDivision on page 1431
TRIGger<m>:LEVel<n>:NOISe:RELative on page 1431

Show trigger lines
Displays the trigger levels and the hysteresis in the diagrams until you disable this option.

Remote command:
DISPlay:TRIGger:LINes on page 1394

Hysteresis transparency
Defines the transparency of the hysteresis area above or below the trigger level.

Only visible if Show trigger lines is enabled.

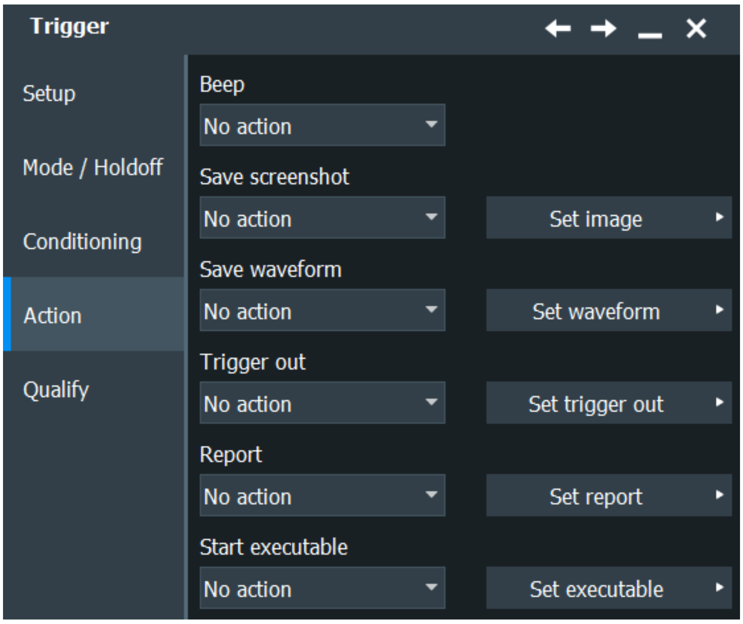
Hysteresis
Displays the automatic hystersis for "Hysteresis mode" = "Auto".

6.6 Actions on trigger

Access: "Menu" > "Trigger" > "Action" tab

The action settings define what happens when a trigger occurs. All available actions can be initiated at the same time.

The R&S RTO can provide an external trigger signal to synchronize the measurements of other instruments. The trigger out signal is also adjusted and enabled in the "Action" tab.



Actions on trigger
The trigger can initiate several actions, each time a trigger occurs. To activate an action, set it to "On trigger". The following actions are available:

"Beep" Generates a beep sound.

"Save screenshot" Saves the waveform data to a file according to settings in "Menu" > "Save/Recall" > "Save" tab > "Screenshot".

"Save Waveform" Saves the waveform data to a file according to settings in "Menu" > "Save/Recall" > "Save" tab > "Waveforms".

"Trigger out" Selects, if a pulse is provided to the [Trigger Out] connector on the rear panel.
A trigger out pulse can be provided either when a trigger occurs, or when a mask test violation occurs, or when a limit check violation in a measurement occurs. See "Trigger out signal setup" on page 228.

"Report" Creates and saves a report using the settings defined in "Menu" key > "Save/Recall" key > "Save" tab > "Report".

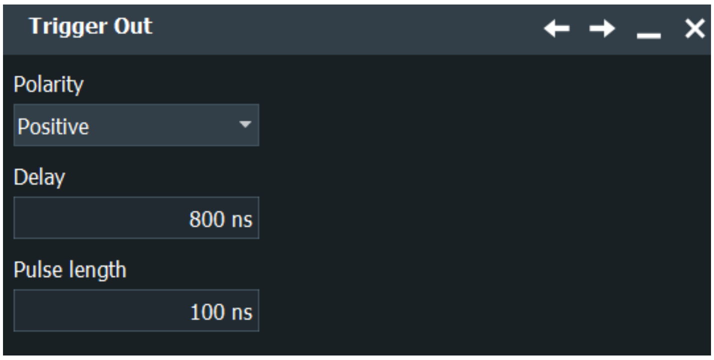
"Start executable" Starts an external application. Tap "Set executable" to set the application path and parameters.
See: Chapter 4.8.3, "External application", on page 113.

Remote command:
TRIGger<m>:EVENt:BEEP on page 1436
TRIGger<m>:EVENt:PRINt on page 1437
TRIGger<m>:EVENt:WFMSave on page 1437
TRIGger<m>:EVENt:RUNexec on page 1437
TRIGger<m>:OUT:ACTion on page 1438

Trigger out signal setup
Available, if "Trigger out" is set to "On trigger".

Defines the pulse that is provided to the [Trigger Out] connector on the rear panel.

A trigger out pulse can be provided either when a trigger occurs, or when a mask test violation occurs, or when a limit check violation in a measurement occurs.

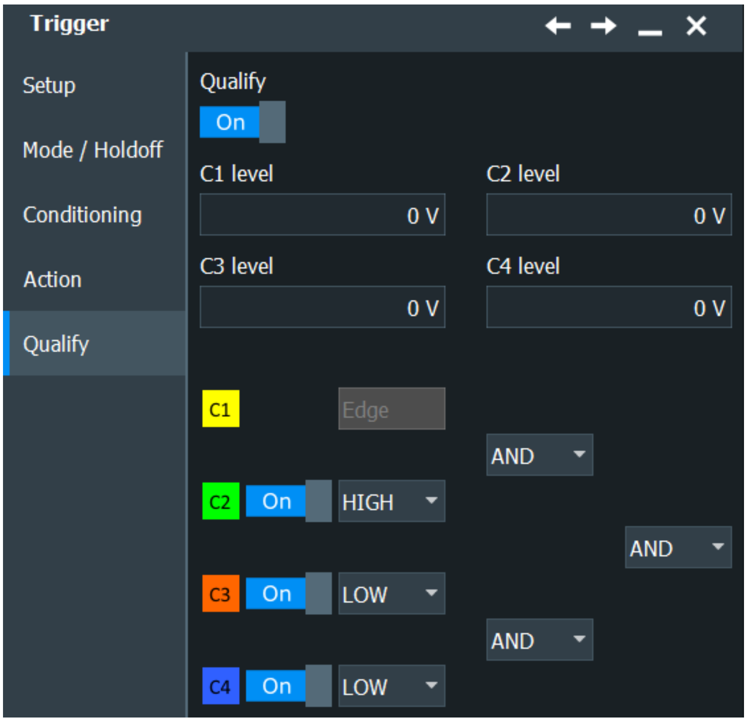


"Polarity"	Sets the polarity of the trigger out pulse, that is the direction of the first pulse edge.
"Delay"	Sets the delay of the first pulse edge to the trigger point. The setting is only available if "Enable trigger out" is active.
"Pulse length"	Sets the length of the trigger out pulse.

Remote command:
`TRIGger<m>:OUT:POLarity` on page 1435
`TRIGger<m>:OUT:PLENgth` on page 1435
`TRIGger<m>:OUT:DELay` on page 1436

6.7 Qualification

Access: "Menu" > "Trigger" > "Qualify" tab



By qualifying a trigger event, you can logically combine the trigger signal with the state of other analog channel signals.

The instrument triggers if both of the following apply:

- The basic conditions of the trigger event definition are fulfilled.
- The logical conditions of the trigger qualification are true.

Qualification is only available for the A-event.

Qualification is not supported if:

- The trigger source is "Extern".
- One of the following trigger types is selected: slew rate, Setup & Hold, Serial pattern, TV, and NFC.

► To enable the qualification settings, select Qualify.

Example: Trigger on write access of a specific device of a bus system

In circuits using SPI, several slave devices use the same lines for reading and writing data, and each slave has its own select line. To trigger on write access of specific slave, the write line is the trigger source and the select line of the slave is set as qualifiying condition.

Qualify

Enables the settings for trigger qualification that are defined in the "Qualification" tab. Qualification adds additional trigger conditions considering the logic states of other digital channel signals.

The checkmark is only active if at least one qualification channel is selected.

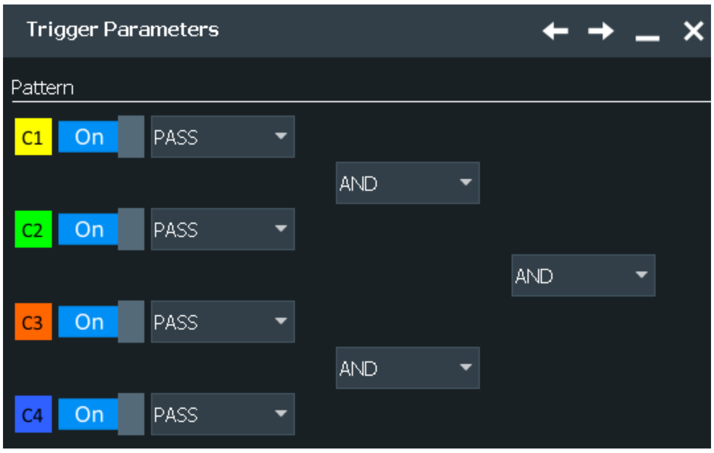
Qualification is available for many trigger types: Edge, Glitch, Width, Runt, Window, Timeout, and Interval.

Qualification is not possible for the R-event.

See also: Chapter 6.7, "Qualification", on page 228

Pattern

The pattern contains the channel selection, and the logical operations structure of hardware-based Boolean logic.



"Channel" Select the channels to be considered. For qualification, you can select all channel signals except for the trigger source. In pattern trigger setup, the trigger source channel is selected by default, and you can select all other channel signals.

"Boolean operator" Defines the logical operation on the digital signal resulting from the comparison with the trigger level.

- "PASS": leaves the input value unchanged
- "NOT": inverts the input value

"Logical operator" Defines the logic combination of two sources. The sources are channel 1/2 and channel 3/4 on the first step, and in the second step the logical combination resulting from the first step.

- "AND": logical AND, conjunctive combination
- "NAND": logical NOT AND
- "OR": logical OR, disjunctive combination
- "NOR": logical NOT OR

Remote command:

`TRIGger<m>:QUALify<n>:A:LOGic` on page 1424
`TRIGger<m>:QUALify<n>:A[:ENABle]` on page 1423
`TRIGger<m>:QUALify<n>:AB:LOGic` on page 1425
`TRIGger<m>:QUALify<n>:ABCD:LOGic` on page 1425
`TRIGger<m>:QUALify<n>:B:LOGic` on page 1424
`TRIGger<m>:QUALify<n>:B[:ENABle]` on page 1423
`TRIGger<m>:QUALify<n>:C:LOGic` on page 1424
`TRIGger<m>:QUALify<n>:C[:ENABle]` on page 1423
`TRIGger<m>:QUALify<n>:CD:LOGic` on page 1425

TRIGger<m>:QUALify<n>:D:LOGic on page 1424
TRIGger<m>:QUALify<n>:D[:ENABle] on page 1423

Trigger Levels
Define the trigger level for each input channel. For state and pattern trigger, the trigger level is a decision threshold: If the signal value is higher than the trigger level, the signal state is high (1 or true for the Boolean logic). Otherwise, the signal state is considered low (0 or false) if the signal value is below the trigger level.

These trigger levels are also used in qualification setup.

You can set the trigger levels for all channels to the same value, see "Couple levels (Trigger level and hysteresis coupling)" on page 212.

Couple levels (Trigger level and hysteresis coupling)
Sets the trigger levels and hysteresis values for all channels to the values of the currently selected trigger source. The function affects only the levels defined for the selected condition. The hysteresis of the external trigger input is an independent value, and it is not affected by level coupling.

In trigger sequences, another coupling of trigger levels is possible: "Couple sequence thresholds" on page 234.

Remote command:
TRIGger<m>:SCOupling on page 1412

6.8 Sequence

A trigger sequence consists of at least one trigger condition and additional conditions defining when the trigger occurs.

The trigger sequence "A → B → R", for example, consists of two subsequent conditions: A-trigger and B-trigger with optional B-trigger delay and count. In addition, a reset condition R can be configured: timeout or R-trigger condition. A-, B-, and R-triggers are configured in the same way.

The instrument checks all trigger settings for compatibility and disables settings that do not fit the previous settings in the sequence.

After the A-trigger conditions have been met, and an optional delay has passed, the B-trigger with independent conditions is enabled. The instrument waits until one or a specified number of B-trigger conditions occur. If the reset condition is not fulfilled, the latest B-trigger causes the trigger event, and then the sequence starts again. The B-trigger can only cause the trigger event if it occurs after the A-trigger and after the delay time.

If you expect, for example, an irregular B-trigger, you can configure a reset condition to restart the sequence. The reset condition can be a simple timeout, and/or a reset event that is defined in the same way as the A- and B-trigger conditions.

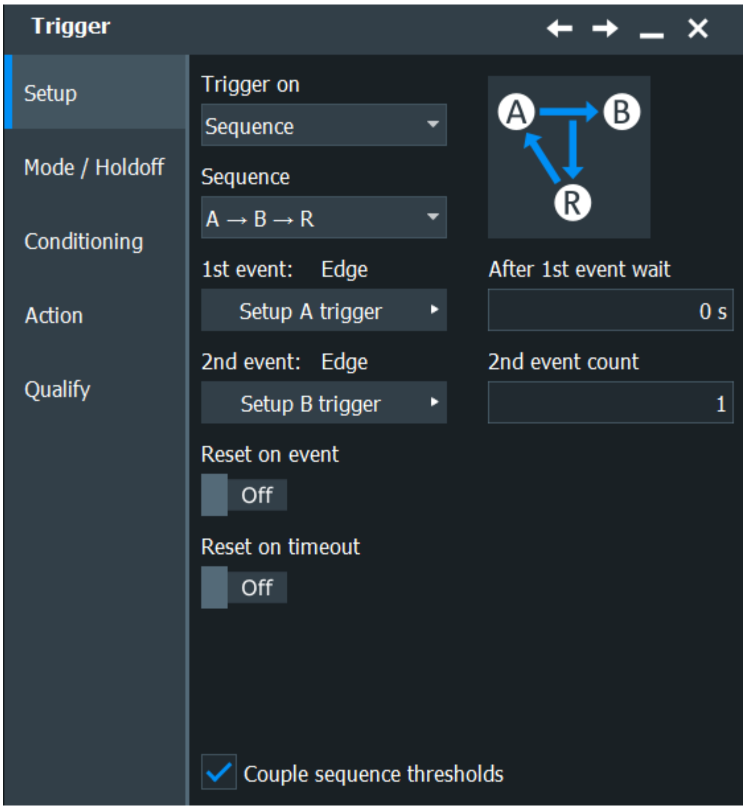
All trigger sequences require that input channels CH1...4 are set as trigger sources for all conditions. All other input sources are disabled. Trigger sequences are not available if one the following trigger types is set as A-trigger:

- Setup & Hold
- TV
- NFC
- CDR

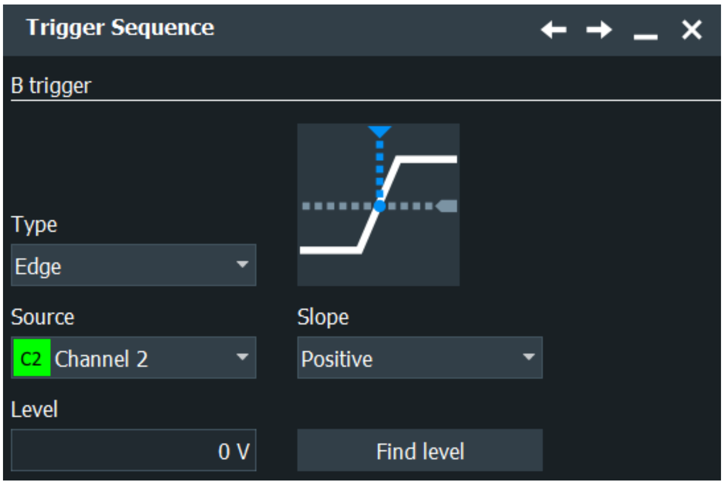
For the zone trigger, more trigger sequences are available, see Chapter 6.9, "Zone trigger", on page 234. Zone trigger requires option R&S RTO-K19.

6.8.1 Sequence setup

Access: "Menu">"Trigger" > "Setup" tab > "Trigger on" = "Sequence"



The B-trigger is the second condition of the trigger sequence. You can configure a delay between the A- and B-trigger, and define a number of fulfilled B-trigger conditions to be ignored. The B-trigger condition is configured in the same way as the A-trigger. The instrument disables settings that do not fit the previous settings in the sequence.



The reset condition R can be a timeout or a trigger condition, or a combination of both.

Sequence

Selects the type of the sequence.

"A → B → R" Triggers if all conditions of A- and B-events, as well as additional delay and reset timeout or R-event (reset) conditions are fulfilled.

"A → Zone" Triggers if the conditions of the A-event and the zone trigger are fulfilled. A holdoff condition also can be set.

"A → B → R → Zone" Triggers if all conditions of A- and B-events, reset timeout or R-event (reset) conditions are fulfilled for the defined zone.

"A OR B" Triggers if the conditions of A-event or the conditions of the B-event are fulfilled.

"A OR B → Zone" Triggers if the conditions of A-event or the conditions of the B-event are fulfilled for the defined zone. A holdoff condition also can be set.

Remote command:
`TRIGger<m>:SEQuence:TYPE` on page 1432

After 1st event wait

Sets the time that the instrument waits after an A-trigger until it recognizes B-triggers.

Remote command:
`TRIGger<m>:SEQuence:DELay` on page 1433

2nd event count

Sets the number of B-trigger conditions to be fulfilled after an A-trigger. The last B-trigger causes the trigger event.

The waiting time for B-triggers can be restricted with a reset condition: timeout or reset event.

Remote command:
`TRIGger<m>:SEQuence:COUNt` on page 1433

Reset on event/Reset event

If enabled, the trigger sequence is restarted by the R-trigger condition if the specified number of B-triggers does not occur before the trigger conditions are fulfilled.

The R-trigger condition is configured in the same way as the A-trigger. The instrument disables settings that do not fit the previous settings in the sequence.

Remote command:
`TRIGger<m>:SEQuence:RESet:EVENt` on page 1433

Reset on timeout/Reset timeout

If timeout is enabled, the instrument waits for the "Timeout" time for the specified number of B-triggers. If no trigger occurs during that time, the sequence is restarted with the A-trigger.

Remote command:
`TRIGger<m>:SEQuence:RESet:TIMeout[:ENABle]` on page 1434
`TRIGger<m>:SEQuence:RESet:TIMeout:TIME` on page 1434

Couple sequence thresholds

Sets the trigger levels to the values of the latest configured trigger condition. Each channel has its own trigger level.

Example:

If you have configured the A-trigger at last, and the trigger level for C1 is 70 mV, enabling the coupling sets the trigger levels for C1 in the B- and R trigger also to 70 mV. If the B-trigger and/or R-trigger use another source as the A-trigger, the level remains unchanged.

Remote command:
`TRIGger<m>:ECOupling` on page 1432

6.9 Zone trigger

The zone trigger function requires option R&S RTO-K19.

The zone trigger triggers on the intersection or non-intersection of the signal and one or more zones or masks. The zone can be applied to any active input signal, math waveform including FFT, and XY-waveform.

You can use the zone trigger, for example, to solve the following tasks:

- Trigger if a peak in the spectrum occurs, define a zone in the FFT diagram to filter amplitude peaks. In the same way, you can filter harmonics.
- Separate rising and falling edges, define a zone around the base or top of the data signal.
- Separate read/write cycles - define a zone in the eye diagram.
- Identify a tube violation of signals with an infrequent non-monotonic edge.
- Filter events in the history, after acquisition

To document the trigger events, use the actions on trigger. For example, create a report when the instrument triggers, or save the waveform.

You can also combine the zone trigger with common trigger conditions and use one of the following trigger sequences:

- "A → Zone"
- "A → B → R → Zone"

Zone trigger is not available for serial protocol triggers.

6.9.1 About trigger zones

A zone is a mask without result box. You can adjust the shape of the zone in the same way as mask segments, graphically by dragging the corner points, or numerically in the "Masks" > "Mask Definition" dialog box. You can also use existing masks in the zone trigger. The zone trigger and usual mask tests run in parallel.

If you switch off a mask test that is used by the zone trigger, the mask is removed from the zone trigger expression automatically.

All zones and masks that are included in the zone trigger expression are indicated with yellow color.

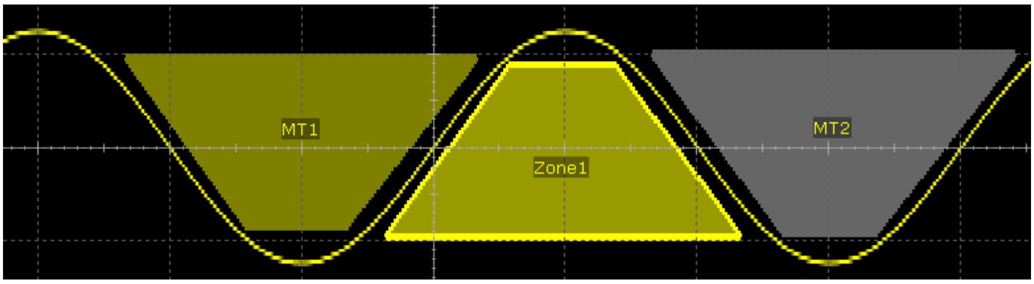


Figure 6-4: Indication of trigger zones

MT1 = Mask is included in zone trigger. Trigger if signal hits the mask (must intersect).
MT2 = Mask is not included in zone trigger.
Zone1 = Zone is included in zone trigger. Trigger if signal does not hit the zone (must not intersect).

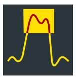
If average or envelope acquisition is enabled, only triggered waveforms are used for envelope and average calculation on channel and math waveforms.

6.9.2 Creating trigger zones

You can define the trigger zones on the display, or use existing mask definitions as trigger zones.

Define trigger zones

1. Tap the "Zone trigger" icon on the toolbar.

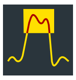
2. In the signal bar, select if the signal must violate the zone to cause a trigger (Must Intersect), or vice versa (Must Not Intersect).
3. Tap the corner points of the zone on the screen.
4. If all points are set, tap "Finish zone" in the signal bar.
   "Zone1" is shown in the expression field.

5. If you want to define another zone, select the logic combination of the zones AND (both zones) or OR (either zone).
6. To create the second zone, repeat steps 3 and 4.
7. To define more zones, repeat steps 5 and 6.

Use existing masks for zone trigger

Prerequisites: At least one mask is created and visible on the display.

1. Tap the "Zone trigger" icon on the toolbar.

2. In the signal bar, select if the signal must violate the zone to cause a trigger (Must Intersect), or vice versa (Must Not Intersect).
3. Tap the mask on the display.
4. Tap "Use selected mask" in the signal bar.
   "MT1" is shown in the expression field.

5. If you want to add another mask test or zone, select the logic combination of the zones AND (both zones) or OR (either zone).
6. To add another mask, tap the mask on the display.
   Tap "Use selected mask" in the signal bar.
7. To add another zone, tap the corner points of the zone on the screen.
   If all points are set, tap "Finish zone" in the signal bar.

If a zone trigger is already defined, you can also add a new mask to the zone trigger when creating the mask.

Intersect
Defines if the signal must intersect the zone to allow the instrument to trigger, or if it must not intersect the zone.

Combine
Sets the logic combination of two zones.

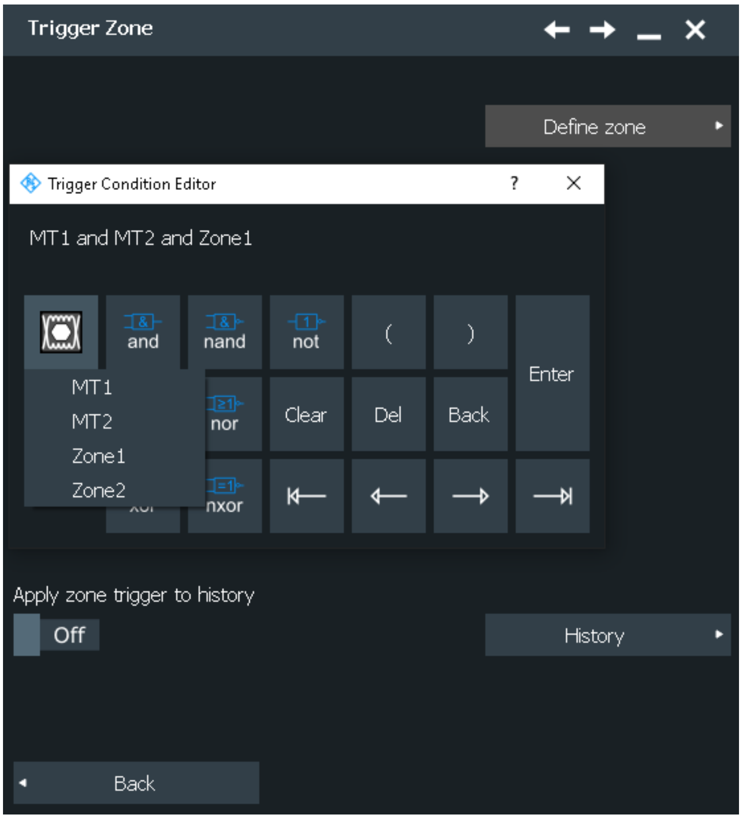
Use selected mask
Includes the selected zone or mask in the zone expression.

Clear Expr
Deletes the zone trigger expression. The zones are not deleted, they remain as usual masks, and the result boxes of the mask test appear.

Delete all
Deletes the zone trigger expression and the zones.

6.9.3 Advanced setup and analysis with zone trigger

Access: toolbar assist for zone trigger > "Advanced Setup", or "Trigger" menu > "Setup" > select sequence with zone trigger > "Setup Z trigger" dialog



Logical trigger condition
If all required mask tests and zones are defined, you can type the logical expression directly, or use the trigger condition editor. All logical combinations are available in the editor. To express the "Must Not Intersect" condition, use the logic NOT.

Remote command:
`TRIGger<m>:ZONE:EXPRession[:DEFine]` on page 1438

Apply zone trigger to history
Applies the zone trigger condition to the acquisitions in the history memory.

The history saves all acquisitions that fulfill the trigger condition that is set during acquisition (zone trigger condition, or another trigger condition). If "Apply zone trigger to history" is disabled, the history "Play" reads and displays all saved acquisitions from the memory.

If the setting is enabled, the zone trigger is applied to the history replay. "Play" displays only acquisitions that fulfill the zone trigger condition.

Using "Apply zone trigger to history", you can:
- Acquire waveforms with high speed and filter them afterwards in the history.
- Change the zone trigger condition after acquisition.

Remote command:
`SWTRigger:HISTory` on page 1438

6.10 External trigger input

Except for using analog or digital input channels as trigger source, you can also use external signals as trigger source. The external signal is connected to the external trigger input, which has a BNC connector. The only trigger type to trigger on external signals is the edge trigger.

1. Connect the external trigger signal to the external trigger input.
2. Set up the trigger:
   a) Select the "Trigger" > "Setup" tab.
   b) Select the source: "Extern"
   c) Adjust the trigger settings.
   See Chapter 6.10.1, "External trigger setup", on page 238.

If the trigger source is a channel input, the trigger system uses the digitized signal. The trigger system of the instrument is a separate system, thus the signal processing by enhancement, decimation and arithmetic has no impact on the trigger signal. Most of the R&S RTO trigger types use the digitized trigger signal.

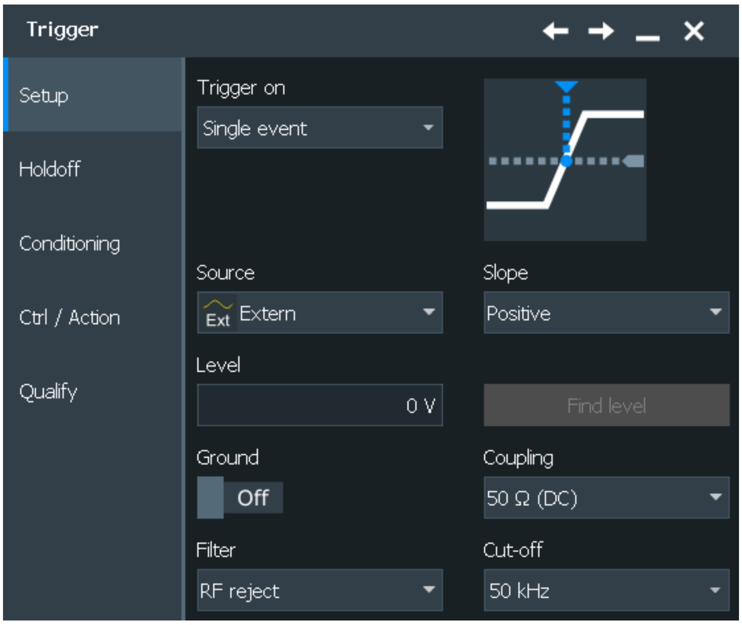
If the trigger source is the external trigger input, the trigger comparator uses the analog input signal. For the external trigger signal, only the edge trigger of the A-trigger is available. Trigger sequence is not supported.

Qualification of the external trigger signal is not available.

6.10.1 External trigger setup

Access: "Menu" > "Trigger" > "Setup" tab > "Source = Extern"

External trigger signals, which are connected to the external trigger input, can be triggered with an edge trigger. The "Find level" function is not available for external trigger signals.



Ground
If the selected trigger source is the external trigger input, you can connect the trigger input to the ground.

Remote command:
TRIGger<m>:ANEDge:GND on page 1397

Coupling
You can set the coupling in the trigger configuration.

"DC 50 Ω"	Direct connection with 50 Ω termination, passes both DC and AC components of the trigger signal.
"DC 1 MΩ"	Direct connection with 1 MΩ termination, passes both DC and AC components of the trigger signal.
"AC"	Connection through capacitor, removes unwanted DC and very low-frequency components.

Remote command:
TRIGger<m>:ANEDge:COUPling on page 1395

Filter
If the selected trigger source is "Extern" (external trigger input), you can directly select a filter to reject high or low frequencies.

For all other trigger sources, you can add a digital filter using the Digital Filter Setup.

"Off"	The trigger signal is not filtered.
"Highpass"	Frequencies below the "Cut-off" frequency are rejected, higher frequencies pass the filter. You can adjust the "Cut-off" frequency, the default is 50 kHz.

"Lowpass" Frequencies higher than the "Cut-off" frequency are rejected, lower frequencies pass the filter.
You can adjust the "Cut-off" frequency, the default is 50 kHz.

Remote command:
`TRIGger<m>:ANEDge:FILTer` on page 1395
`TRIGger<m>:ANEDge:CUToff:HIGHpass` on page 1396
`TRIGger<m>:ANEDge:CUToff:LOWPass` on page 1396

Slope

Sets the edge type for the trigger condition.

"Positive" Selects the rising edge, that is a positive voltage change.

"Negative" Selects the falling edge, that is a negative voltage change.

"Both" Selects the rising as well as the falling edge. This option is not available if the trigger source is the external trigger input.

Remote command:
`TRIGger<m>:EDGE:SLOPe` on page 1395
`TRIGger<m>:ANEDge:SLOPe` on page 1397
`TRIGger<m>:SLEW:SLOPe` on page 1408

Trigger level

Sets the voltage level for the trigger condition. You can also drag the trigger level marker on the display (TA or TB on the right edge of the display). The range of the trigger level is limited in a way so that always a hysteresis for stable trigger conditions is available.

Remote command:
`TRIGger<m>:LEVel<n>[:VALue]` on page 1393

7 Waveform analysis

This chapter describes general methods to check and analyze waveforms. These are:

7.1 Zoom

The zoom functions allow you to magnify a specific section of the diagram in order to view more details. You can define several zoom areas for the same diagram and even couple them, or you use the hardware zoom.

7.1.1 Methods of zooming

The R&S RTO provides various ways of zooming: You define the section of a diagram that you want to magnify, and the zoomed view is shown in a separate zoom diagram. Additionally, you can magnify the diagram directly: The hardware zoom changes the horizontal and vertical scales of the diagram so that you see the selected section.

There are different ways to initiate and configure the zoom function:

- **Fingertip zoom**: magnifies the waveforms around your fingertip. When you drag your finger, the magnifier moves, too. You can convert the fingertip zoom into a standard zoom diagram.
- **Graphical method**: you draw, move and adjust the zoom area on the touchscreen – a very quick and simple method for standard zoom and hardware zoom.
- **Numeric method**: you enter x- and y-values in a dialog box or adjust them using navigation controls. These are precise ways which can be used to optimize a graphically defined zoom.
  With the numeric method there are two ways of defining the zoom area:
  – Specifying **start and stop values** for the x- and y-axes; the acquired data within those values is zoomed.

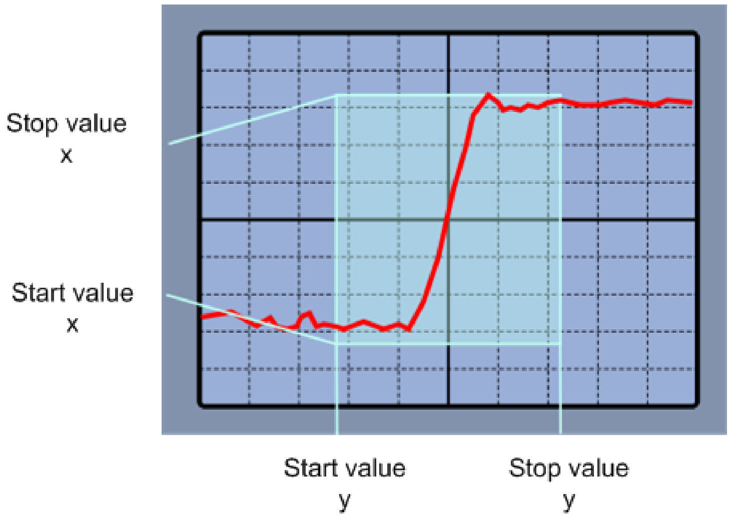


Figure 7-1: Numeric zoom using start and stop values

– Specifying the x- and y-**position** of the centerpoint of the area plus a **range** for the x- and y-axes; the area defined by that centerpoint and the ranges is zoomed.

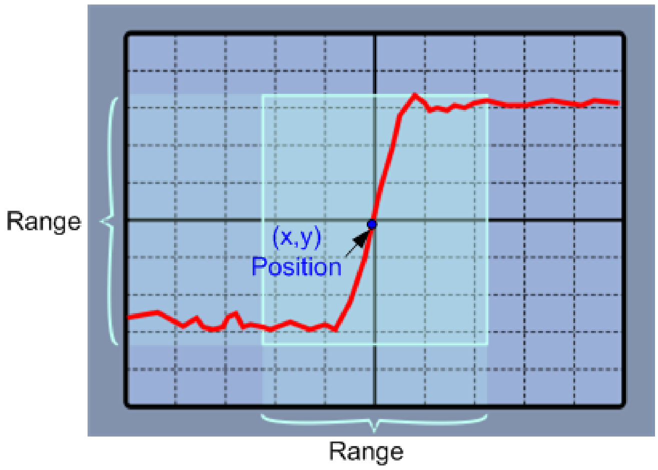


Figure 7-2: Numeric zoom using position and range

- **Coupled zoom** creates a copy of the selected zoom area. Coupled zoom areas always have the same size (size coupling). They can be positioned separately or together (position coupling).

Zoom areas can be used for gating, for example, to define a measurement gate. You can set the gate exactly to the limits of the zoom.

Evaluation gates - available histogram areas, masks, and measurement gates - can be displayed in zoom diagrams to simplify the graphical gate adjustment on the touch-screen.

See: Chapter 7.1.3, "Zooming for details", on page 247.

7.1.2 Zoom settings

The zoom area, i.e. the section to be enlarged, can be defined using two different methods:

- Using the zoom functions on the toolbar and draw the zoom area on the touch-screen
- Specifying numeric values:
  - start and stop values for the x- and y-axes
  - x and y position of one point in the diagram plus a range for the x- and y-axes

  See also: Chapter 7.1.1, "Methods of zooming", on page 241.

7.1.2.1 Zoom functions on the toolbar

The zoom icon on the toolbar shows the last selected zoom type. A short tap on the icon activates the selected zoom. If you touch the icon and drag your finger down, a menu opens where you can select another zoom type.

Standard zoom
Displays a magnified section of the diagram in an additional zoom diagram. It is a display zoom, instrument settings are not changed.

Touch and hold the zoom area to open the "Zoom" dialog box.

Remote command:
`LAYout:ZOOM:ADD` on page 1440

Hardware zoom
Changes the instrument settings - horizontal and vertical scales as well as trigger level and offset - to display a part of the diagram in greater detail.

Coupled zoom
Creates a coupled zoom area and its related zoom diagram. If you change the size of one zoom area, the size of all coupled zoom areas is changed as well.

Remote command:
`LAYout:ZOOM:ADDCoupled` on page 1440

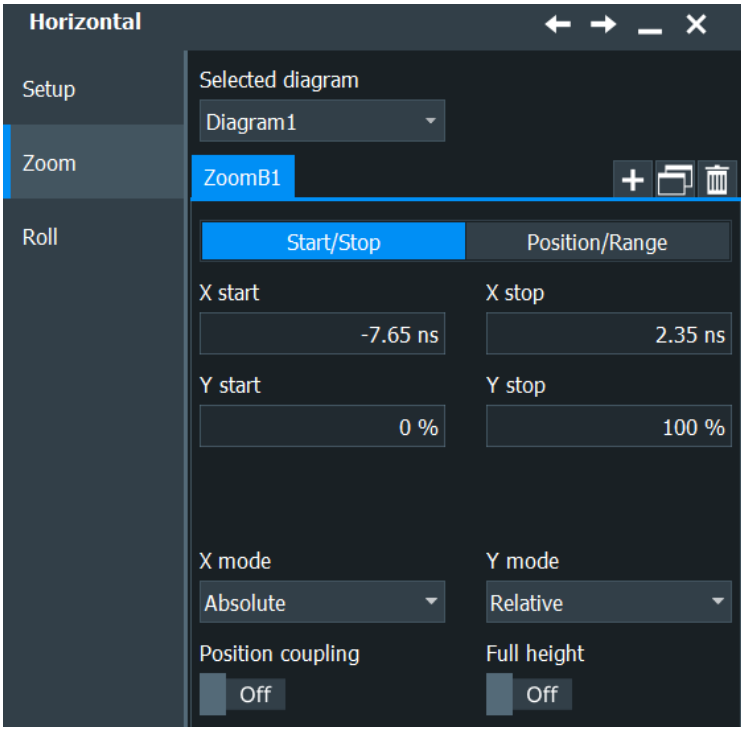
Fingertip zoom
Magnifies the waveforms around your fingertip.

Tap the icon and put your finger on the waveform. The touched part of the waveform is displayed in a magnifier. Drag your finger on the screen to move the magnifier. You can change the zoom factor using the [Navigation] knob.

7.1.2.2 Zoom settings

Access: "Menu" > "Horizontal" > "Zoom" tab.

The "Zoom" tab allows you to specify start and stop values for the x- and y-axes. The acquired data within these ranges is zoomed.

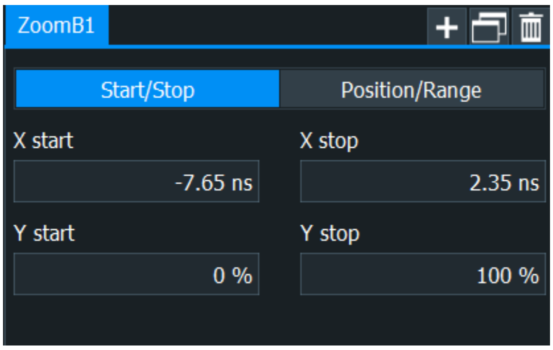


Additionally, you can enable a "Zoom Overlay" in the "Menu" > "Settings" > "Appearance" > "Diagram", see Chapter 4.3.4, "Diagram appearance settings", on page 84.

Selected diagram
Indicates which of the waveform diagrams is selected for zooming.

Mode
Selects how the window for the zoom diagram is defined. You can select between defining "Start/Stop" values or "Position/Range".

Start/Stop
The "Start/Stop" tab allows you to specify start and stop values for the x- and y-axes. The acquired data within these ranges is zoomed.



X start ← Start/Stop
Defines the lower limit of the zoom area on the x-axis.

Remote command:
`LAYout:ZOOM:HORZ:ABSolute:STARt` on page 1442
`LAYout:ZOOM:HORZ:RELative:STARt` on page 1444

X stop ← Start/Stop
Defines the upper limit of the zoom area on the x-axis.

Remote command:
`LAYout:ZOOM:HORZ:ABSolute:STOP` on page 1443
`LAYout:ZOOM:HORZ:RELative:STOP` on page 1444

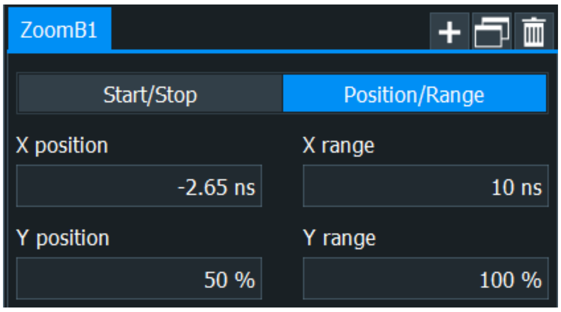
Y start ← Start/Stop
Defines the lower limit of the zoom area on the y-axis.

Remote command:
`LAYout:ZOOM:VERTical:RELative:STARt` on page 1447
`LAYout:ZOOM:VERTical:ABSolute:STARt` on page 1446

Y stop ← Start/Stop
Defines the upper limit of the zoom area on the y-axis.

Remote command:
`LAYout:ZOOM:VERTical:RELative:STOP` on page 1447
`LAYout:ZOOM:VERTical:ABSolute:STOP` on page 1446

Position/Range
In the "Position/Range" tab, you specify the x and y position of center point of the zoom area plus a range for the x- and y-axes; the area defined by that point and the ranges is zoomed.



X position ← Position/Range
Defines the x-value of the centerpoint of the zoom area.

Remote command:
`LAYout:ZOOM:HORZ:ABSolute:POSition` on page 1441
`LAYout:ZOOM:HORZ:RELative:POSition` on page 1443
`SEARch:RESDiagram:HORZ:ABSolute:POSition` on page 1632
`SEARch:RESDiagram:HORZ:RELative:POSition` on page 1633

X range ← Position/Range
Defines the width of the zoom area.

Remote command:
LAYout:ZOOM:HORZ:ABSolute:SPAN on page 1442
LAYout:ZOOM:HORZ:RELative:SPAN on page 1443
SEARch:RESDiagram:HORZ:ABSolute:SPAN on page 1633
SEARch:RESDiagram:HORZ:RELative:SPAN on page 1633

Y position ← Position/Range
Defines the y-value of the centerpoint of the zoom area.

Remote command:
LAYout:ZOOM:VERTical:ABSolute:POSition on page 1445
LAYout:ZOOM:VERTical:RELative:POSition on page 1446
SEARch:RESDiagram:VERT:ABSolute:POSition on page 1634
SEARch:RESDiagram:VERT:RELative:POSition on page 1635

Y range ← Position/Range
Defines the height of the zoom area.

Remote command:
LAYout:ZOOM:VERTical:RELative:SPAN on page 1447
LAYout:ZOOM:VERTical:ABSolute:SPAN on page 1445
SEARch:RESDiagram:VERT:ABSolute:SPAN on page 1634
SEARch:RESDiagram:VERT:RELative:SPAN on page 1635

X mode
Defines whether absolute or relative values are used to specify the x-axis values.

Remote command:
LAYout:ZOOM:HORZ:MODE on page 1441
SEARch:RESDiagram:HORZ:MODE on page 1633

Y mode
Defines whether absolute or relative values are used to specify the y-axis values.

Remote command:
LAYout:ZOOM:VERTical:MODE on page 1444
SEARch:RESDiagram:VERT:MODE on page 1635

Position coupling
Enables or disables the position coupling of coupled zooms. If position coupling is enabled and you move one zoom area, the other coupled zoom areas are moved, too, and keep their distance.

Remote command:
LAYout:ZOOM:POSCoupling on page 1441

Full height
Uses the full diagram height for the zoom area. Only horizontal zoom settings can be changed.

7.1.3 Zooming for details

The usage of the various zoom methods is described in the following procedures:

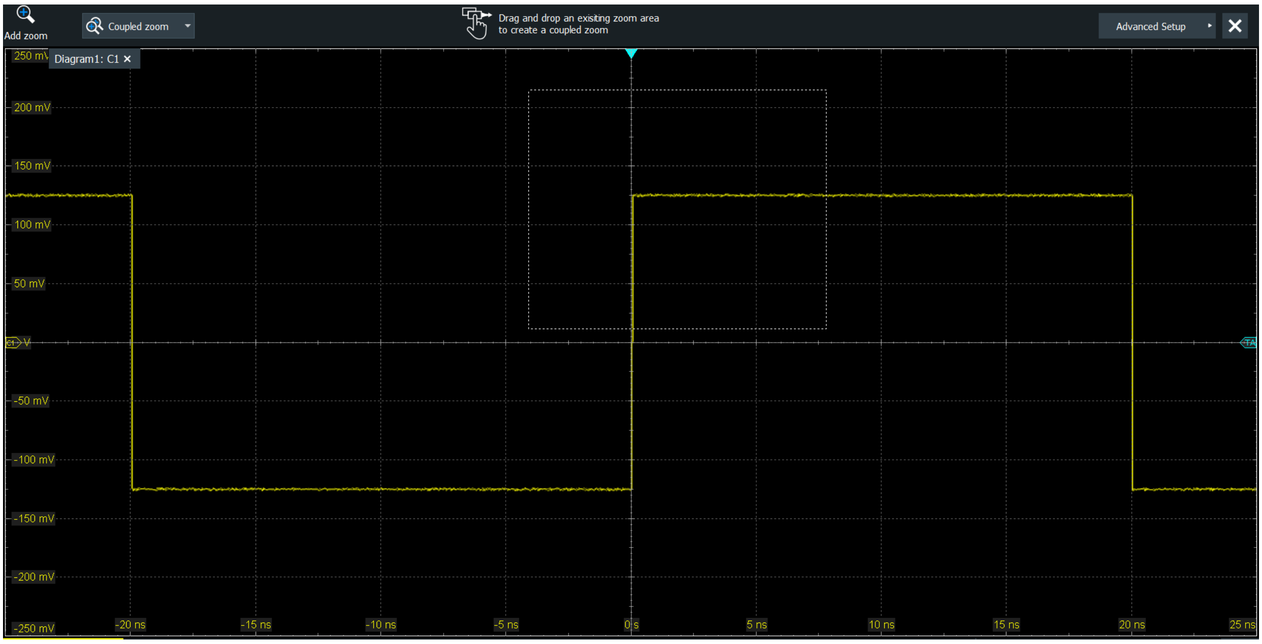
- To define the zoom area graphically on the touchscreen
- To define the zoom area numerically using start/stop values
- To define the zoom area numerically using position and range values
- To define multiple zoom areas
- To define coupled zoom areas

To define the zoom area graphically on the touchscreen

For graphical zooming, you use your finger on the screen.

1. On the toolbar, tap the "Standard Zoom" icon.

2. Touch the position that you want to define as one corner of the zoom area. Then drag your finger to the opposite corner of the zoom area.
While you drag your finger on the touchscreen, a dotted rectangle indicates the current zoom area. When the rectangle covers the required zoom area, remove your finger.



The indicated area is magnified in a new zoom diagram. The original diagram is displayed with the zoom area indicated as a rectangle.

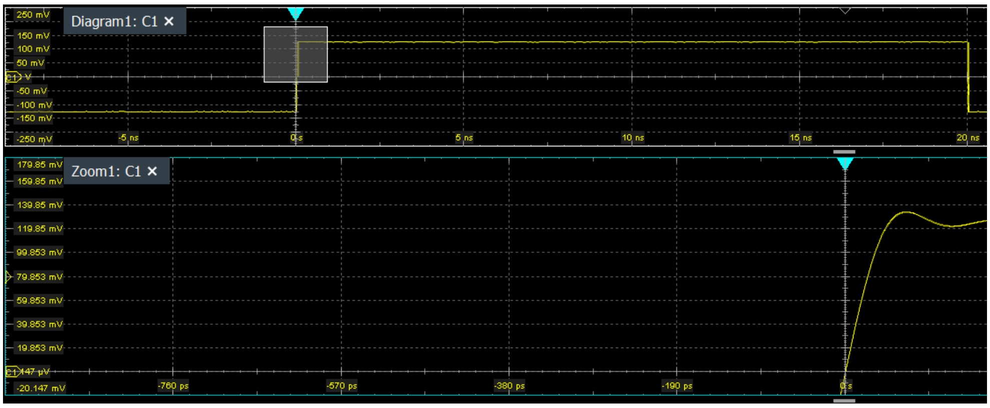


Figure 7-3: Zoom diagram and overview diagram

3. If the position of the zoom area is not correct, drag the rectangle in the overview to the correct position.
4. If the size of the zoom area is not yet ideal, tap the rectangle in the overview diagram.
   Now, 4 red lines indicate the edges of the zoom area. A dashed red line indicates the selected edge, which you can adjust.
5. Touch the edge that you want to move, and drag it to the required position.

To optimize the zoom definition of an active zoom diagram, double-tap the zoom diagram. The "Zoom" dialog box for numeric definition is opened.

To create a new zoom using the Zoom dialog box

1. There are two ways to create a new zoom:
   - If you want to create a new, unconfigured zoom, tap the "Add" icon.
     
   - If you want to create zoom based on an existing one, tap the "Copy" icon.
     
2. Enter a name for the zoom using the on-screen keyboard.

To define the zoom area numerically using start/stop values

1. On the "Menu" menu, tap "Horizontal".
2. In the "Zoom" tab, select "Start/Stop".
3. Select "X mode" to define "Absolute" or "Relative" x-axis values. Relative values cause the zoom area to adapt to the input values dynamically.
4. Define the "X start" and "X stop" values that define the lower and upper borders of the zoom area on the x-axis (see Figure 7-1).

5. Select the "Y mode" to define "Absolute" or "Relative" y-axis values.
6. Define the "Y start" and "Y stop" values that define the lower and upper borders (respectively) of the zoom area on the y-axis.

   When you close the dialog box, the specified area is magnified in a new zoom diagram. The original diagram is displayed with the zoom area indicated as a rectangle (see Figure 7-3).

To define the zoom area numerically using position and range values

1. On the "Menu" menu, tap "Horizontal".
2. In the "Zoom" tab, select "Position/Range".
3. Select "X mode" to define "Absolute" or "Relative" x-axis values. Relative values cause the zoom area to adapt to the input values dynamically.
4. Under "X position", define the x-value of the center point of the zoom area (see Figure 7-2).
5. Under "X range", define the width of the zoom area.
6. Select the "Y mode" to define "Absolute" or "Relative" y-axis values.
7. Under "Y position", define the y-value of the center point of the zoom area.
8. Under "Y range", define the height of the zoom area.

   When you close the dialog box, the specified area is magnified in a new zoom diagram. The original diagram is displayed with the zoom area indicated as a rectangle.

To define multiple zoom areas

You can define more than one zoom area for the same diagram, for example to compare several peaks in a measurement. These zoom areas can be displayed in separate zoom diagrams, or together in one zoom diagram.

To define multiple zoom areas graphically, simply repeat the steps described in To define the zoom area graphically on the touchscreen - for each area. Numerically, proceed as follows:

1. On the "Menu" menu, tap "Horizontal".
2. Select the "Zoom" tab.
3. Select the required tab according to the method you want to use to define the zoom area.
4. To copy the current zoom area definition, tap the "Copy" icon.
   Alternatively, tap the "Add" icon to add a new zoom area.
5. Enter a name for the new zoom diagram using the displayed on-screen keyboard.
6. Define the zoom area as described for the first zoom.

An additional zoom diagram is displayed for the new zoom area, and another rectangle in the original diagram indicates the new zoom area. Each rectangle in the overview has the same color as the corresponding zoom diagram frame.

7. Alternatively, you can overlay the zoom areas: The zooms are shown in the same zoom diagram, as if the zoom areas are overlaid:
   In the "Settings" > "Appearance" dialog > "Diagram " tab , enable "Zoom > Overlay".

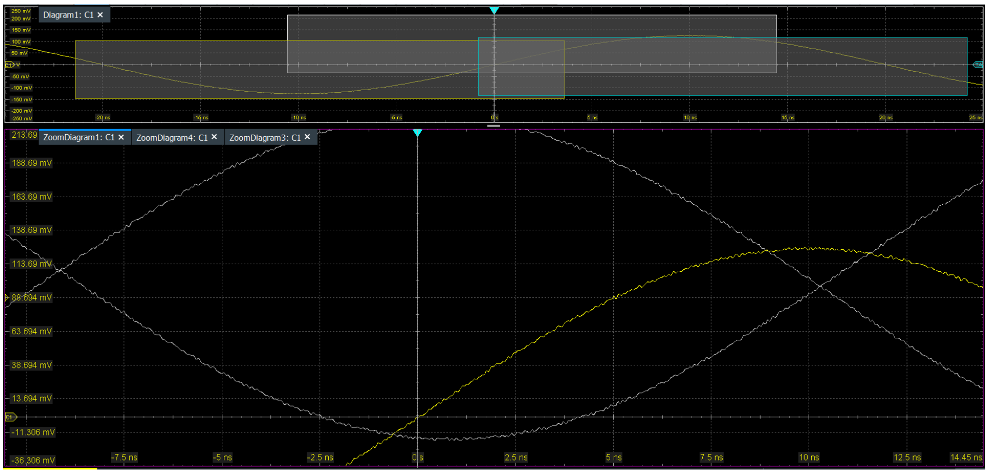


Figure 7-4: Multiple zoom diagrams. Left: separate zoom diagrams. Right: overlaid zoom

To define coupled zoom areas

You can define multiple zoom areas for one diagram that are coupled. If you change the size of one zoom area, the size of all coupled zoom areas is changed as well. Furthermore, you can couple also the position in order to move all coupled zooms at once. Coupling is useful, for example, if you want to compare recurring peaks in a signal.

1. On the toolbar, tap the "Coupled Zoom" icon.

2. In the diagram overview, tap an existing zoom area.

   The selected zoom area is duplicated.

3. Drag the duplicate zoom area to the required position.
4. To create further coupled zooms, repeat the steps above.

   Now, if you change the zoom area size of any of the coupled zoom areas in the "Zoom" dialog box, the settings are changed for all coupled zoom areas.

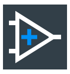
5. In the "Zoom" dialog box, select the diagram that contains the coupled zooms.
6. Select a zoom tab.
7. Enable "Position coupling".

If you move one of the coupled zoom areas in the diagram, all other coupled zooms are moved as well, and their distance is kept unchanged.

To use the hardware zoom

In contrast to the normal zoom, the hardware zoom changes the instrument settings - horizontal and vertical scales, and also the trigger level and offset. Thus, the selected area is displayed in the diagram instead of the original waveform. No additional zoom diagram is opened.

1. On the toolbar, tap the "Hardware Zoom" icon.

2. Drag your finger on the touch screen to mark the zoom area.
   A dotted rectangle indicates the current zoom area. When the rectangle covers the required zoom area, remove your finger. The diagram changes and shows the magnified area.

   Tip: To return to the previous display, use the "Undo" icon.

   Note: You can combine hardware zoom and normal zoom - first use the hardware zoom, then the zoom into the display. The reverse approach is also possible: Create a zoom diagram, and then apply the hardware zoom to the waveform diagram. Both the waveform and the zoom diagrams are changed.

7.2 Reference waveforms

You can configure up to four reference waveforms to display stored waveforms. Any active signal or mathematical waveform can be stored as a reference waveform. It can then be loaded again later to restore the waveform on the screen.

7.2.1 Working with reference waveforms

Reference waveforms can be displayed in addition to the signal waveforms, saved to file, and loaded back for further analysis. Reference waveforms can be loaded only from BIN files.

Note: Saving and loading reference waveforms, and preset with active reference waveform delete the undo stack. After these actions, undo is not possible.

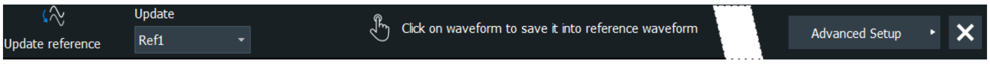
To update a reference waveform using the toolbar icon

If you often need to update a reference waveform, you can use the "Save Reference" toolbar icon.

1. Add the "Save Reference" icon to the toolbar, see Chapter 3.3.6.2, "Configuring the toolbar", on page 57 .

2. Tap the icon.

   The toolbar assist opens.



3. Set "Update" to the reference waveform to be used.
4. Tap the waveform to be used as reference waveform.

To display a reference waveform

1. In the "Menu" > "Apps" > "Analysis" tab, select "Reference Curve".
   Alternatively, press the [Ref] key.
2. Select the tab for the reference waveform you want to display ("R1"-"R4").
3. Load a stored reference waveform as described in "To load a reference waveform" on page 253, or select a source to be displayed as a reference:
   a) In the "Setup" tab, select "Signal".
   b) Select the "Source" from the selection list. The source can be any active signal, math, or other reference waveform.
   c) Tap the "Update" button to update the current reference waveform with the source data.
4. Tap the "Show" button.

   The reference waveform is displayed on the screen.
5. A reference waveform can have its own scaling settings or it can be scaled according to the source settings. By default, the scaling of the reference waveform is coupled to the source settings. Additionally, it can be stretched or compressed in vertical and horizontal direction.
   If necessary, change the settings on the "Vertical" and "Rescale" tabs of the "Reference Waveform" dialog boxes.
   To restore the original settings, tap the "Restore settings" in the "Vertical" tab.
   For a description of the scaling settings, see Chapter 7.2.2.4, "Rescale", on page 257

To save a reference waveform

1. In the "Save/Recall" dialog, select "Save > Waveform".

   Tip: You can also save a waveform as a reference waveform in the "File" dialog box, see Chapter 12.2.6, "Saving and loading waveform data", on page 469. Here, you can also save multiple waveforms in one file.
2. Select the tab for the reference waveform you want to store ("Ref1"-"Ref4").
3. Display and configure the reference waveform as described in "To display a reference waveform" on page 252.

4. Select the file format.

   Note: Reference waveforms can be loaded only from BIN files. XML and CSV formats are meant for further processing in other applications.

5. To save the waveform to the currently selected file, tap "Save". By default, the prefix for reference waveform files is "RefCurve".
   To save the waveform to another file, select "Save As".
   Enter a file name and select the directory. The file type is already defined according to the selection in the previous step. In order to load the reference waveform on the instrument again later, use the file type BIN.

   The source settings of the reference waveform and the current scaling settings are stored to the specified file.

To load a reference waveform

Note: Reference waveforms can be loaded only from BIN files.

1. Press the [Ref] key.
2. Select the tab for the reference waveform you want to load ("Ref1" - "Ref4").
3. In the "Setup" tab, select "File".
4. To load the waveform from the specified file, tap "Open".
   To load the waveform from a different file, tap "Open". Select the file from the file selection dialog box. Only BIN files are displayed in the file list.

   The selected waveform is loaded as the specified reference waveform.
   If multiple waveforms are saved in the file, you are asked to assign each waveform to a reference waveform. All waveforms are loaded together.

To view a reference waveform

► You can view a reference waveform, using the "Saveset with preview" function.
  For details, see: Chapter 12.1.4, "Saveset with preview function", on page 446.

7.2.2 Settings for reference waveforms

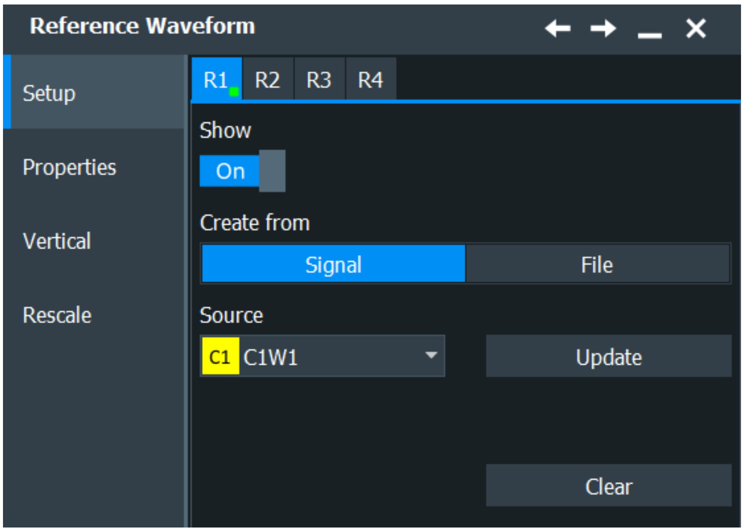
To compare waveforms and analyze differences between waveforms, you can use up to four reference waveforms R1 to R4. Each reference waveform has its own memory on the instrument. You can also save an unlimited number of reference waveforms and load them for further use.

The display of a reference waveform is independent from the display of the source waveform; you can move, stretch and compress the curve vertically and horizontally.

7.2.2.1 Reference waveform setup

Access: [Ref] key > "Setup" tab

In the "Setup" tab, you select the reference waveform and its source. The source can be an active waveform - trace of an input channel, math waveform or another reference waveform - or a stored waveform.



R1/2/3/4

Each tab contains the settings for one of the four available reference waveforms.

Show

Displays the reference waveform in the diagram.

Remote command:

`REFCurve<m>:STATe` on page 1449

Signal

Selects a signal as the reference waveform.

Source ← Signal

Selects the source waveform from the active waveforms of input channels, math signals and other reference waveforms.

Remote command:

`REFCurve<m>:SOURce` on page 1448

Update ← Signal

Copies the selected source waveform with all its settings to the memory of the reference waveform. If the acquisition is running, the reference waveform is a snapshot.

Remote command:
REFCurve<m>:UPDate on page 1450

File
Opens a file selection dialog box and loads the selected reference waveform file.

Double-tap the filename to open the file selection dialog box, see also Chapter 12.5, "File selection dialog", on page 478.

Note: Note that reference waveforms can be loaded from .bin files only. xml and csv formats are meant for further processing in other applications.

Remote command:
REFCurve<m>:OPEN on page 1449
REFCurve<m>:SAVE on page 1450
REFCurve<m>:DELete on page 1450

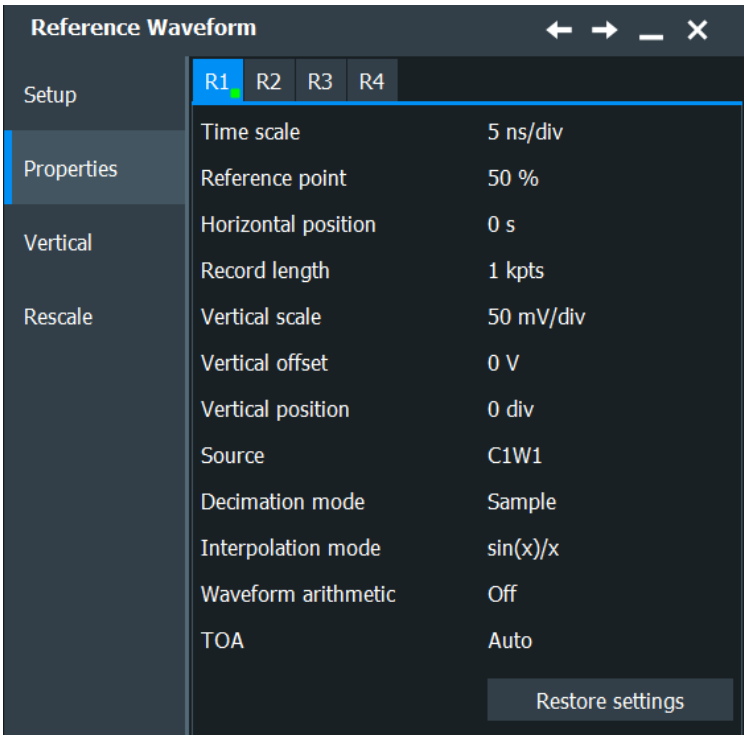
Clear
The selected reference waveform disappears, its memory is deleted.

Remote command:
REFCurve<m>:CLEar on page 1450

7.2.2.2 Properties

Access: [Ref] key > "Properties" tab

A reference waveform can be scaled, stretched and positioned in the diagram. The "Original Attributes" tab shows the original settings of the reference waveform, which are stored together with the waveform data.

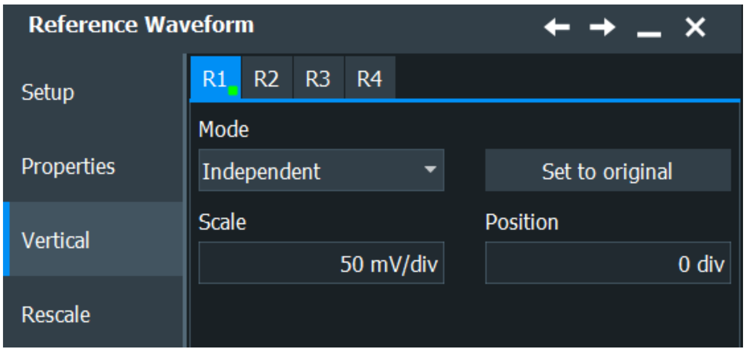


Restore Settings
Resets the time scale and the reference point to the original values of the reference waveform.

Remote command:
REFCurve<m>:RESTore on page 1451

7.2.2.3 Vertical

Access: [Ref] key > "Vertical" tab



Mode
Selects the type of vertical settings:

"Coupled to source" Vertical position and scale of the source are used.

"Independent" Scaling and position can be set specific to the reference waveform.

Remote command:
REFCurve<m>:VMODe on page 1451

Set to original
Restores the original vertical settings of the reference waveform (vertical scale, position, and offset), if vertical scaling is set to "Independent".

Remote command:
REFCurve<m>:TOORignal on page 1452

Scale
Sets the vertical scale for the reference waveform, if vertical scaling is set to "Independent". You can also use the vertical [Scale] knob to adjust this value.

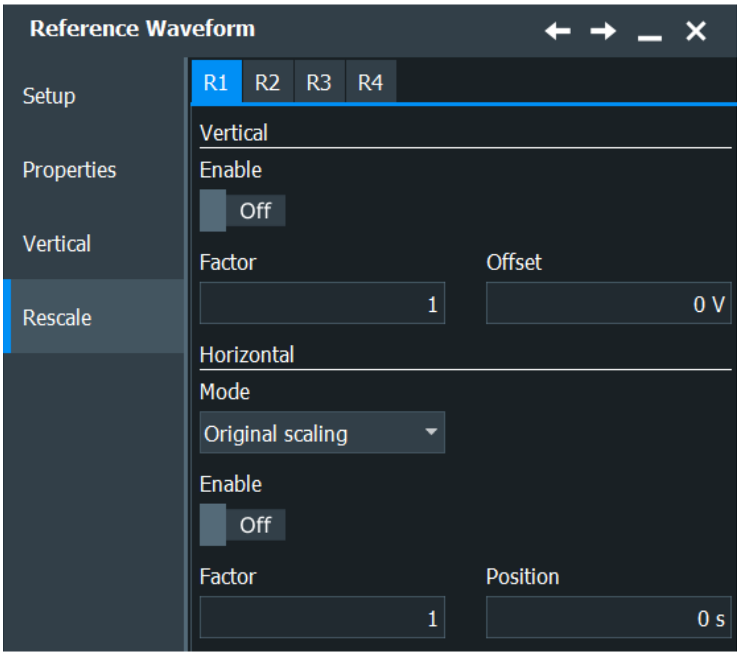
Remote command:
REFCurve<m>:SCALe on page 1451

Position
Moves the reference waveform up or down in the diagram, if vertical scaling is set to "Independent". If "Position" is assigned to the vertical [Position] knob, you can also use this knob to adjust the "Vertical position"

Remote command:
REFCurve<m>:POSition on page 1452

7.2.2.4 Rescale

Access: [Ref] key > "Rescale" tab



Vertical Rescaling
Stretching and offset change the display of the waveform independent of the vertical scale and position.

Enable ← Vertical Rescaling
If enabled, the vertical offset and factor are applied to the reference waveform.

Remote command:
`REFCurve<m>:RESCale:VERTical:STATe` on page 1452

Factor ← Vertical Rescaling
A factor greater than 1 stretches the waveform vertically, a factor lower than 1 compresses the curve.

Remote command:
`REFCurve<m>:RESCale:VERTical:FACTor` on page 1452

Offset ← Vertical Rescaling
Moves the reference waveform vertically. Enter a value with the unit of the waveform. Like vertical offset of a channel waveform, the offset of a reference waveform is subtracted from the measured value. Negative values shift the waveform up, positive values shift it down.

Note: As for all waveforms, a vertical offset of a reference waveform can be set using the vertical [Position] knob. This offset is independent from the reference scaling offset, which is described here. If both offsets are set, their values are added up.

Remote command:
`REFCurve<m>:RESCale:VERTical:OFFSet` on page 1453

Mode ← Vertical Rescaling
Selects the type of horizontal settings:

"Adjust to X Axis"	The current horizontal settings of the diagram are used.
"Original Scaling"	Horizontal scaling and reference point of the source waveform are used.

Remote command:
REFCurve<m>:HMODe on page 1453

Horizontal Rescaling ← Vertical Rescaling
Stretching and offset change the display of the waveform independent of the horizontal settings of the source waveform and of the horizontal diagram settings.

Enable ← Horizontal Rescaling ← Vertical Rescaling
If enabled, the horizontal offset and factor are applied to the reference waveform.

Remote command:
REFCurve<m>:RESCale:HORizontal:STATe on page 1454

Factor ← Horizontal Rescaling ← Vertical Rescaling
A factor greater than 1 stretches the waveform horizontally, a factor lower than 1 compresses the curve.

Remote command:
REFCurve<m>:RESCale:HORizontal:FACTor on page 1454

Position ← Horizontal Rescaling ← Vertical Rescaling
Moves the waveform horizontally. Enter a value with a time unit suitable for the time scale of the diagram. Positive values shift the waveform to the right, negative values shift it to the left.

Remote command:
REFCurve<m>:RESCale:HORizontal:POSition on page 1455

7.3 Mathematics

Math waveforms are calculated waveforms. You can define up to eight math waveforms and display them on the screen, and use it as source for further analysis.

Math waveforms are defined by mathematical expressions (formulas). You can enter mathematical expressions using different methods:

- "Operator" tab: you define a simple math function in a graphical editor by selecting the source waveforms and the operator.
- "Equation" tab: you define sophisticated math functions in a formula editor, as required to your needs.

The result of an FFT analysis is a specific math waveform. For information on FFT and spectrograms, see Chapter 9, "Spectrum analysis", on page 371.

The vertical scale of a math waveform is adapted automatically to the measurement results to ensure optimal display. Furthermore, you can scale each math waveform manually in vertical direction like a channel waveform.

As for channel waveforms, you can also change the arithmetic mode for the waveform to display the envelope or an average over several calculations.

You can store a math waveform as a reference waveform and restore it later, see "To save a reference waveform" on page 252.

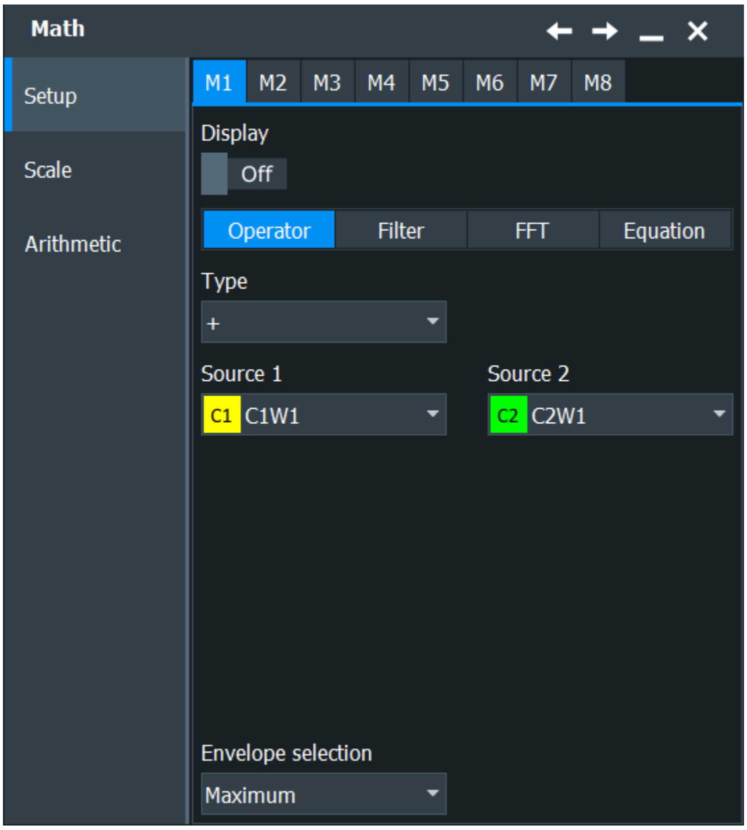
7.3.1 Displaying math waveforms

Math waveforms can be displayed in addition to the channel and other waveforms. They also can be used for analysis, e.g. measurements, even if the math waveform is not active.

1. In the "Math" menu, select "Math Setup".
   Alternatively, press the [Math] key.
2. Define the math expression for calculation in one of the following ways:
   - Chapter 7.3.3.2, "Defining a formula in the Operator editor", on page 262
   - Chapter 7.3.4, "Equation", on page 263
   - Chapter 9.1.2, "Configuring spectrum waveforms", on page 375
3. In the "Math Setup" dialog box, in the "Setup" tab, tap the "Enable math signal".
   The math waveform is displayed on the screen.
4. To change the vertical scaling of the math waveform, tap the "Manual" icon.
5. Enter the "Vertical scale" factor (per division). If necessary, add a "Vertical offset".
   By default the instrument performs an automatic scaling.
   Tip: You can also use the vertical [Scale] rotary knob for scaling. In this case, the scale mode is set to "Manual" temporarily.
6. If you need the envelope or average of the math waveform over several calculations, change the arithmetic mode for the waveform as for channel waveforms.
   See also: "Wfm Arithmetic" on page 136.
7. Close the "Math Setup" dialog box.

7.3.2 Math setup - general settings

You can define up to eight different math waveforms. Each waveform is defined in a separate tab in the "Math" dialog box ("M1" to "M8").



The settings for input of mathematical formulas are described in separate chapters:

- Chapter 7.3.3.1, "Settings in the Operator editor", on page 261
- Chapter 7.3.5, "Filters", on page 271
- Chapter 9.1.3, "FFT configuration settings", on page 379
- Chapter 7.3.4, "Equation", on page 263

Display
If activated, a diagram for the defined math waveform is displayed on the touch screen.

Remote command:
`CALCulate:MATH<m>:STATe` on page 1458

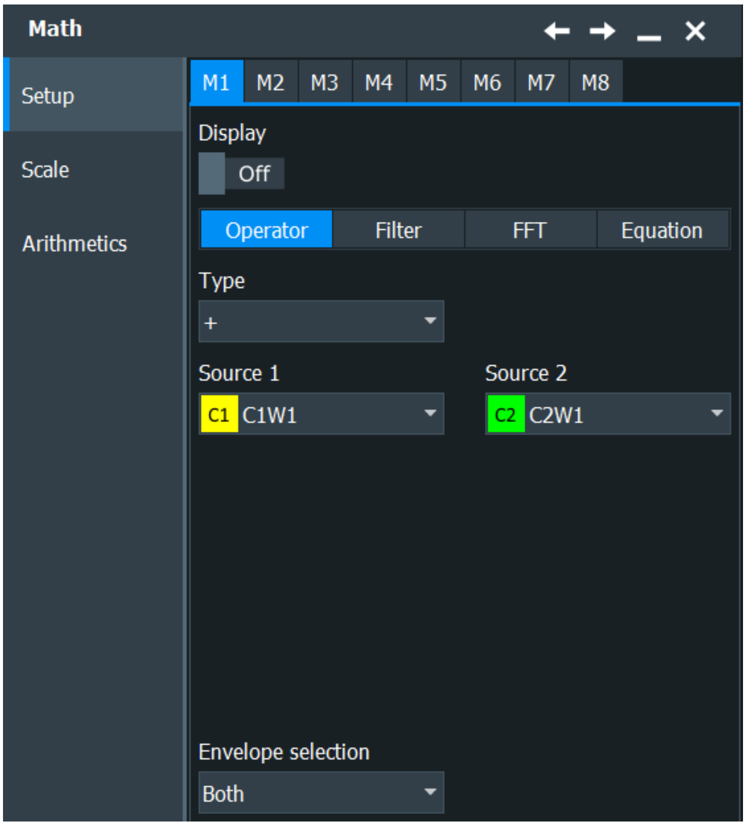
Display
Selects the upper or lower part of the waveform for mathematic calculation, or a combination of both.

The setting is relevant for waveforms with waveform arithmetic mode "Envelope" or with "Peak detect" decimation. All mathematic operations - except for derivation - can be applied to envelope waveforms and waveforms with "Peak detect" decimation.

Remote command:
`CALCulate:MATH<m>:ENVSelection` on page 1458

7.3.3 Operator editor

In theOperator editor, you can define the most common mathematical formulas without knowing their correct syntax.



Remote command:

- `CALCulate:MATH<m>[:EXPRession][:DEFine]` on page 1458

7.3.3.1 Settings in the Operator editor

Source 1 / 2
Defines the signal source to be evaluated by the math function. Waveform 1 of channel signals can be selected.

Note: If you require other signal sources not listed here, use the formula editor provided in the Equation tab. Any waveform of any input channel can be used as a source. See: Chapter 7.3.4, "Equation", on page 263.

Type
Defines the type of operation to be performed on the selected signal sources. The following functions are available:

Note: If you require other operators not listed here, use the formula editor provided in the "Equation" tab. See: Chapter 7.3.4, "Equation", on page 263.

"+"	Adds up the sources
"-"	Subtracts source 2 from source 1.
"x"	Multiplies source 1 by source 2.
"|x|"	Determines the absolute value of the source.
"dx/dt"	Differentiates the source value with respect to the time value. Not possible on envelope waveforms and waveforms with "Peak detect" decimation.
"log(x)"	Calculates the logarithm of the source value based on 10.
"ln(x)"	Calculates the natural logarithm of the source value (based on e).
"ld(x)"	Calculates the binary logarithm of the source value (binary logarithm, based on 2).
"Rescale"	Rescales the source values by a factor "a" and an offset "b": *ax+b*. See also: "a / b" on page 262.

Noise reject
Only available for "Type" = "dx/dt".

Sets the number of neighboring samples that are skipped for differentiation.

To suppress noise effects during differentiation, it can be useful not to consider two directly neighboring points to calculate *dx* (x_n-x_{n-1}). Instead, some samples in-between are skipped and a point a few samples further is used (e.g. x_n-x_{n-3}).

a / b
Defines the values for the "Rescale" function (ax+b).

"a"	Is the factor the signal source is multiplied with.
"b"	Is the offset of the signal source on the y-axis.

7.3.3.2 Defining a formula in the Operator editor

1. Open the "Menu" > "Math" dialog.
   Alternatively, press the [Math] key.
2. In the "Setup" tab, select the "Operator" tab.

3. Tap the "Source 1" and "Source 2" icons and select the signal sources to which the math function is applied. For details on available signal sources, see "Source 1 / 2" on page 262.
4. Tap the "Type" icon and select the mathematical function.
   For details on available operators, see "Type" on page 262.
5. If the operator requires additional parameters, enter them in the input fields.

7.3.4 Equation

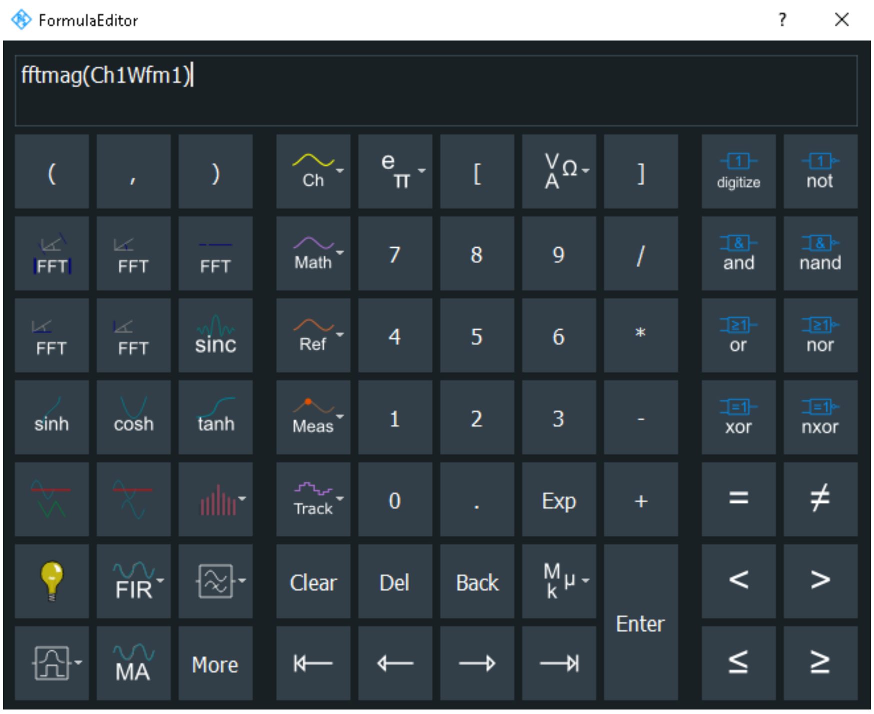
In the Equation tab, you can enter complex formulas to define a math waveform. The formula editor helps to enter formulas easily with correct syntax, using a large selection of operators and signal sources.

All formulas in the formula editor are linear regardless whether a [dB] is set or not.

► Double-tap the Equation tab to display the formula editor.

7.3.4.1 Formula editor

Using the formula editor you can define math functions freely, using a large selection of operators and signal sources. For a procedure on using the editor, see Chapter 7.3.4.11, "Defining a formula in the formula editor", on page 270.



Remote command:

- CALCulate:MATH<m>[:EXPRession][:DEFine] on page 1458

Buttons of the formula editor and their usage

Table 7-1: Basics

Icon	Description	Usage/Comment, *FormulaEditor expression*
(	left bracket	enclose operands
,	comma	separates operands
)	right bracket	enclose operands
e / π	math. constants	e: Euler number: 2.7182... Pi: 3.1415...
[	left square bracket	enclose unit
V / A / Ω	units	[<unit>]
]	right square bracket	enclose unit
x^a	exponentiation with base x	x: base, a: exponent *x^a*
/	division	
*	multiplication	
-	subtraction	
+	addition	
0...9	numeric characters	
.	decimal point	
Exp	exponentiation with base 10	*e*
Enter	expression complete	insert expression in "Setup" dialog and close the formula editor
Clear	clear expression in editor	restart editing
Del	Delete	remove selected part of expression
Back	Backspace	remove last symbol, operator or operand to the left of the cursor
M / k / µ	SI-prefix for unit	<SI-prefix>[<unit>]

Table 7-2: Signal sources

Icon	Description	Usage/Comment, *FormulaEditor expression*
Ch	signal waveform	*Ch<1...4>Wfm<1...3>*
Math	math waveform	*Math<1...8>*
Ref	reference waveform	*Ref<1...4>*

Icon	Description	Usage/Comment, *FormulaEditor expression*
Meas	measurement waveform	*Meas<1...8>*
Track	track waveform	*Track<1...8>*

Table 7-3: Cursor keys

Icon	Description	Usage/Comment, *FormulaEditor expression*
|←	move cursor to beginning	
←	move cursor 1 step to the left	
→	move cursor 1 step to the right	
→|	move cursor to end	

7.3.4.2 Math functions: algebra

Table 7-4: Algebra

Icon	Description	Usage/Comment, *FormulaEditor expression*
$\|x\|$	absolute x value	*abs(x)*
$\sqrt{x}$	square root of x	*sqrt(x)*
x^2	x*x	*pow(x)*
$\log_{10}$	common logarithm (base 10)	*log(x)*
$\log_e$	natural logarithm (base e)	*ln(x)*
$\log_2$	binary logarithm (base 2)	*ld(x)*
e^x	exponentiation with base e	*exp(x)*
$\int x dx$	integral of x	*integral(x)*
d/dx	derivation of x	*derivation(x,y)* with x = waveform and y = number of skipped sam-ples (noise reject)
ax+b	scaling of x	*rescale(x,a,b)*

7.3.4.3 Math functions: trigonometry

Table 7-5: Trigonometry (More keys)

Icon	Description	Usage/Comment, *FormulaEditor expression*
sinh	hyperbolic sine	*sinh(x)*
cosh	hyperbolic cosine	*cosh(x)*
tanh	hyperbolic tangent	*tanh(x)*

7.3.4.4 Math functions: bit operations

Table 7-6: Bit operations

Icon	**Description**	**Usage/Comment, *FormulaEditor expression***
digitize	convert to 0 or 1	*digitize(x)*
not	negation	*not(x)*
and		*and*
nand	negation of and	*nand*
or		*or*
nor	negation of or	*nor*
xor	exclusive or	*xor*
nxor	negation of exclusive or	*nxor*

7.3.4.5 Math functions: comparison

Table 7-7: Comparison

Icon	**Description**	**Usage/Comment, *FormulaEditor expression***
=	equal	=
≠≠	not equal	<>
<	smaller	<
>	greater	>
≤	smaller or equal	<=
≥	greater or equal	>=
More	display additional keys	

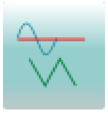
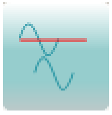
7.3.4.6 Math functions: FFT

Table 7-8: FFT (More keys)

Icon	**Description**	**Usage/Comment, *FormulaEditor expression***
|FFT|	magnitude of FFT value	*fftmag(x)*
FFT (φ)	FFT phase value	*fftphi(x)*
FFT -dφ*df	FFT group delay	*fftgroupdelay(x)*
FFT (re)	real part of FFT value	*fftre(x)*
FFT (im)	imag part of FFT value	*fftim(x)*

7.3.4.7 Math functions: correlation

Table 7-9: Correlation (More keys)

Icon	Description	Usage/Comment, *FormulaEditor expression*
	Cross correlation function of two waveforms Measures the similarity of two waveforms as a function of a time-lag applied to one of them. Function limits the maximum record length to 4 MSa. Two modes of normalization are supported: biased and unbiased. The length of the correlation buffer is $N_0 + N_1 - 1$ samples. The length of the first input signal is N_0 samples and the length of the second signal is N_1 samples.	*correlation(x1, x2, biased)* *correlation(x1, x2, unbiased)* with x1 = waveform 1 and x2 = waveform 2 *correlation(x1, x2)* performs an unbiased correlation
	Auto correlation Used to find repeating patterns, for example, a periodic signal obscured by noise. The length of the auto correlation buffer is $2N - 1$ samples, if the length of the input signal is N samples. Two modes of normalization are supported: biased and unbiased.	*autocorrelation(x, biased)* *autocorrelation(x, unbiased)* with x = channel waveform *autocorrelation(x)* performs an unbiased autocorrelation
	biased / unbiased normalization for correlation and auto correlation	see above

Mathematic background for correlation:

$$Temp1\_R_{xy}(m) = \sum_{n=0}^{N1} y_n^* x_{n+m} \quad m \in [0; N_1[$$

$$Temp0\_R_{xy}(m) = \sum_{n=1}^{N0} x_n^* y_{n+m} \quad m \in [1; N_0[$$

The R&S RTO uses only the real part of the signal. Two modes of normalization are supported: biased and unbiased.

$$R_{xy}(m) = \begin{cases} \dfrac{1}{\min(N_0, N_1)} Temp1\_R_{xy}(m) & m \in [N_0 - 1; N_1 + N_0 - 1[\\ \dfrac{1}{\min(N_0, N_1)} Temp0\_R_{xy}^*(-m) & m \in [0; N_0 - 1[\end{cases}$$

Equation 7-1: Biased correlation

$$R_{xy}(m) = \begin{cases} \dfrac{1}{a(m)} Temp1\_R_{xy}(m) & m \in [N_0 - 1; N_1 + N_0 - 1[\\ \dfrac{1}{a(m)} Temp0\_R_{xy}^*(-m) & m \in [0; N_0 - 1[\end{cases}$$

Equation 7-2: Unbiased correlation

Mathematic background for auto correlation:

$$R_{xx}(m) = \begin{cases} \frac{1}{N} \sum_{n=0}^{N-m-1} x_n^* x_{n+m} & m \geq 0 \\ R_{xx}^*(-m) & m < 0 \end{cases}$$

Equation 7-3: Biased auto correlation

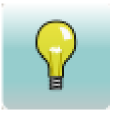
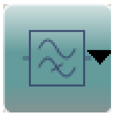
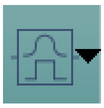
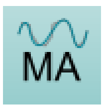
$$R_{xx}(m) = \begin{cases} \frac{1}{N-|m|} \sum_{n=0}^{N-m-1} x_n^* x_{n+m} & m \geq 0 \\ R_{xx}^*(-m) & m < 0 \end{cases}$$

Equation 7-4: Unbiased auto correlation

The R&S RTO uses only the real part of the signal.

7.3.4.8 Math functions: filter and power

Table 7-10: Filter and power (More keys)

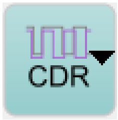
Icon	Description	Usage, comment, *FormulaEditor expression*
	Electric power	Electric power is calculated from voltage, based on measurement impedance (see Impedance) *elecpower(x)* = U^2/R
	Finite impulse response (FIR) filter	*FIR(tpye,source,limit,shape)* or *FIR(userdef,source,path)* See Chapter 7.3.5.2, "FIR filter in the formula editor", on page 272
	Type of FIR filter	*highpass* or *lowpass*, see FIR filter
	Characteristics of FIR filter	*gaussian* or *rectangle*, see FIR filter
	Moving average	Calculates a mean value of several adjacent sample points. The result is a smoothed waveform. The moving average uses the full data and can be used for non-periodic signals. It works like a lowpass filter and increases the vertical resolution at the expense of bandwidth reduction. *MovingAverage(x,y)* with: x = source (channels only), y = number of samples to be averaged Example: *MovingAverage(Ch1Wfm1,1000)* Averages 1000 subsequent samples of the channel 1 waveform

7.3.4.9 Math functions: CDR

The clock data recovery function requires at least one of the options R&S RTO-K12, or R&S RTO-K133, or R&S RTO-K134.

Hardware CDR requires option R&S RTO--K13.

Table 7-11: Clock data recovery (More keys)

Icon	Description	Usage/Comment, *FormulaEditor expression*
	CDR: displays the generated clock signal as math waveform	Hardware: *CDR(hw)* Software 1: *CDR(sw1,x)* Software 2: *CDR(sw2,x)* where x is the signal from which the clock is recovered Example: *CDR(sw1,CH2Wfm1)* See also: Chapter 18.3.3, "Displaying the recovered clock signal", on page 1117

7.3.4.10 Transfer functions of a step-like signal

The R&S RTO provides two functions to calculate the transfer function of a system out of its step response. These functions are not supported by the formula editor, but you can enter them directly in the "Advanced" tab in the "Math Setup" dialog box. To enter the formula, you need a keyboard (onscreen keyboard or a connected usual one).

Math expression	Description
Step2FreqRespNormMag(x,points) Where: • x is the waveform (channel, math, reference) • points is the number of points in the resulting math waveform (min. 1000 points)	Normalized magnitude of transfer function in frequency domain Example: *Step2FreqRespNormMag(Ch1,2000)*
Step2FreqRespNormPhi(x,points,time offset) Where: • x is the waveform (channel, math, reference) • points is the number of points in the resulting math waveform (min. 1000 points) • time offset in s	Normalized phase of transfer function in frequency domain Example: *Step2FreqRespNormPhi(Ch1,2000,2e-9)*

The calculation of transfer functions has several steps:

- Calculate the frequency domain representation of a step-like signal. A detailed description is given in: A. M. Nicolson, „Forming the fast Fourier Transform of a step response in Time-Domain Metrology,“ in Electronic letters, Bd. 9, Nr. 14, p. 317, 1973.
- Calculate the derivative to convert the result to the transfer function.
- Normalize the result at DC to a magnitude of 1 V and subtract specified time offset from phase.

The step-like signal to be analyzed is a time domain waveform, and the result is a math waveform in the frequency domain.

To get useful results, check and adjust the following settings:

1. Place the analyzed step of the signal in the left half of the time domain diagram. Use the "Reference point" or the horizontal "Position" of the trigger to move the signal.
   See also: Chapter 5.4, "Acquire settings", on page 131.
2. Adjust the record lengths of the source signal and the math signal. The "Record length" of the step-like source signal must be shorter than the number of points in the resulting math function, which is set in the formula. See also: "Record length" on page 132.
3. If the resulting math waveform is noisy, increase the "Average count" for the source signal until the waveform is clear.
   See also: Chapter 5.4.2, "Mode settings", on page 134.

7.3.4.11 Defining a formula in the formula editor

1. Open the "Menu" > "Math" dialog.
2. Select the Equation tab.
3. Double-tap the editing area.
   The "Formula Editor" is displayed.
4. Enter the math formula including all required signal sources and operators by selecting the corresponding keys in the editor. For details on the available keys, see Chapter 7.3.4.1, "Formula editor", on page 263.
5. To insert a physical unit in the formula, proceed as follows:
   a) If necessary, insert a decimal prefix using the "M/k/µ" key.
   b) Insert an opening square bracket using the "[" key.
   c) Insert the physical unit using the "V/A/Ω" key.
   d) Insert a closing square bracket using the "]" key.
   The resulting expression could be, for example: `m[V]`
6. To perform a rescaling function, proceed as follows:
   a) Select the rescaling function using the "ax+b" key.
   b) Behind the left bracket, insert the signal source using one of the following keys:
      - "Ch" for a channel
      - "Math" for a math function
      - "Ref" for a reference waveform
      - "Meas" for a measurement
   c) Insert a comma using the "," key.
   d) Insert the "a" value, i.e. the scaling factor, using the number keys.
   e) Insert a comma using the "," key.
   f) Insert the "b" value, i.e. the scaling offset, using the number keys.
   g) Insert the closing bracket using the ")" key.

The resulting expression could be, for example: `rescale(Ch1Wfm1,3,4)`

7.3.5 Filters

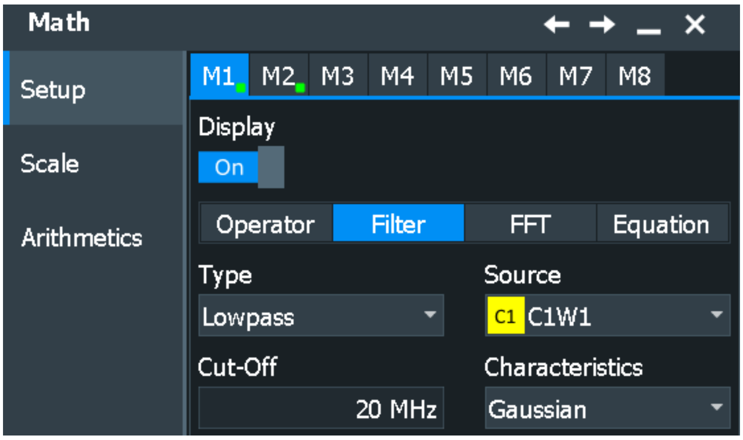
The R&S RTO provides several ways to filter the input signal:

- Lowpass filter by selecting a bandwidth limit in the acquisition path (vertical channel settings)
  See "Analog" on page 146
- Digital lowpass filter
  See Chapter 5.5.5, "Digital filter setup", on page 148
- FIR filter to create a filtered math waveform (lowpass, highpass, bandpass)
  You can set up a FIR-filtered math waveform in the basic math editor, or in the advanced editor. Both ways are described below in this chapter.

7.3.5.1 FIR filter in the math setup

Access: "Math" > "Setup" tab > "Filter" tab.

The finite impulse response filter is a filter to create filtered waveforms with lowpass, highpass, or bandpass. The filter requires additional settings.



- "Type": defines whether the FIR filter is a highpass, lowpass, or bandpass/bandstop (= "User defined") filter.
- "Cut-Off": sets the limit frequency for the highpass or lowpass FIR filter.
- "Characteristics": relevant for lowpass filter. Defines whether it has a Gaussian or a rectangular shape. The highpass is always Gaussian.
- "Select filter file": relevant for bandpass/bandstop filter. Opens a file dialog to select the file with the filter coefficients.

Cut-off frequency for lowpass FIR filter

The cut-off frequency depends on the horizontal resolution and the filter characteristics. The frequency for the lowpass filter can only be set in this range:

f_g_3dB = (0.001 ... 0.2) f_a_in* for Gaussian FIR filter

f_g_3dB = (0,001 ... 0.4) f_a_in* for rectangular FIR filter

Where: *f_g_3dB* = cut-off frequency to be set for the lowpass filter, and *f_a_in* = reciprocal of the resolution, or sample rate.

Cut-off frequency for highpass FIR filter

To check limit frequency for the highpass filter, convert it to an equivalent lowpass frequency:

f_LP = f_a_in/2 - f_HP

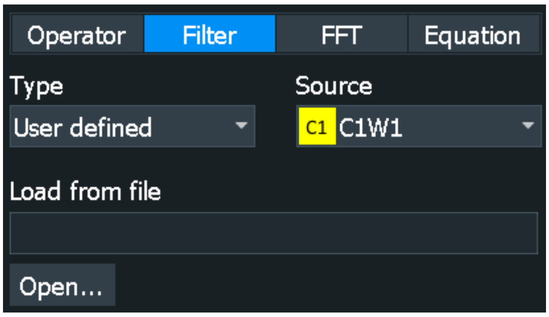
Where f_HP is the requested highpass limit frequency and f_LP the equivalent lowpass frequency that has to comply with the limits given above.

Bandpass and bandstop FIR filter: CSV file

To define a bandpass or bandstop, you need a CSV file that contains the comma-separated filter coefficients. The maximum number of filter coefficients is 2 048 000 taps. No other parameters are allowed in the file.

To create the CSV file, we recommend using the Matlab Filter Design & Analysis tool. In the tool, enter the filter type, filter order and filter frequencies. Make sure to set the sample frequency in Matlab and the sample rate at the oscilloscope to the same value. If the values differ, the filter is shifted in frequency.

In R&S RTO, in the "Math" > "Setup" > "Filter", select the filter "Type" = "User defined", and load the CSV file.



7.3.5.2 FIR filter in the formula editor

You can type the FIR filter formula directly in the "Equation" tab, or open the formula editor and use the buttons to create the formula.

Highpass and lowpass FIR filter

The general syntax for highpass and lowpass filters is:

FIR(tpye,source,limit,shape)

- Type is *lowpass* or *highpass*
- Source is the input channel of the signal.
- Limit is the cut-off frequency

- Shape is *gaussian* or *rectangle* for the lowpass filter, and *gaussian* for the highpass.

► For example, to set a rectangle lowpass filter on channel 1 with 10 MHz cut-off frequency, enter:
FIR(lowpass,Ch1,1e+07,rectangle)

► For example, to set a Gaussian highpass filter on channel 2 with 3 GHz cut-off frequency, enter:
FIR(highpass,Ch2,3e+09,gaussian)

Bandpass and bandstop FIR filter

The syntax for the bandpass filter is:

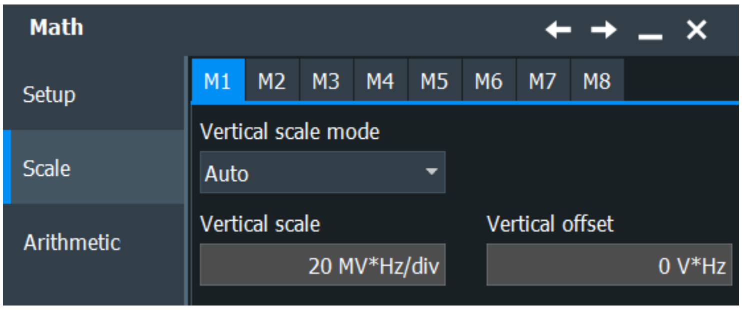
FIR(userdef,source,path)

- Source is the input channel of the signal.
- Path is a string containing the path and filename of the filter file. The file contains the comma-separated filter coefficients.

► For example, to set a bandpass on channel 1 with filter coefficients saved in the `bandpass.csv` file, enter:
FIR(userdef,Ch1,"C:\Users\Public\Documents\Rohde-Schwarz\RTx\bandpass.csv")

7.3.6 Math scale settings

Access: [Math] > "Scale" tab.



In this tab, you can find functions to set the vertical parameters of the math waveform. Additional settings are available for FFT expressions, see FFT scale settings.

7.3.6.1 General settings

Vertical scaling mode (Manual/Auto)
By default, the vertical scale is adapted to the current measurement results automatically to provide an optimal display. However, if necessary, you can define scaling values manually to suit your requirements.

Note: When you change the scaling values manually using the "Scale" rotary knob, the scale mode is set to "Manual" temporarily. When you edit the math function, scaling is automatically set back to "Auto" mode. "Manual" mode is only maintained during math function changes if you select it yourself.

"Manual"	Enter the required values for "Vertical scale" and "Vertical offset". For FFT, set "Vertical range" and "Vertical maximum".
"Auto"	"Vertical scale" and "Vertical offset" are read-only. For FFT, only the "Vertical maximum" is read-only.

Vertical Scale

Defines the scale of the y-axis in the math function diagram. The value is defined as "<unit> per division", e.g. *50m V/div*. In this case, the horizontal grid lines are displayed in intervals of 50 mV.

If the "Vertical scaling mode (Manual/Auto)" on page 273 is set to "Auto", this setting is read-only.

Remote command:
`CALCulate:MATH<m>:VERTical:SCALe` on page 1461

Vertical Offset

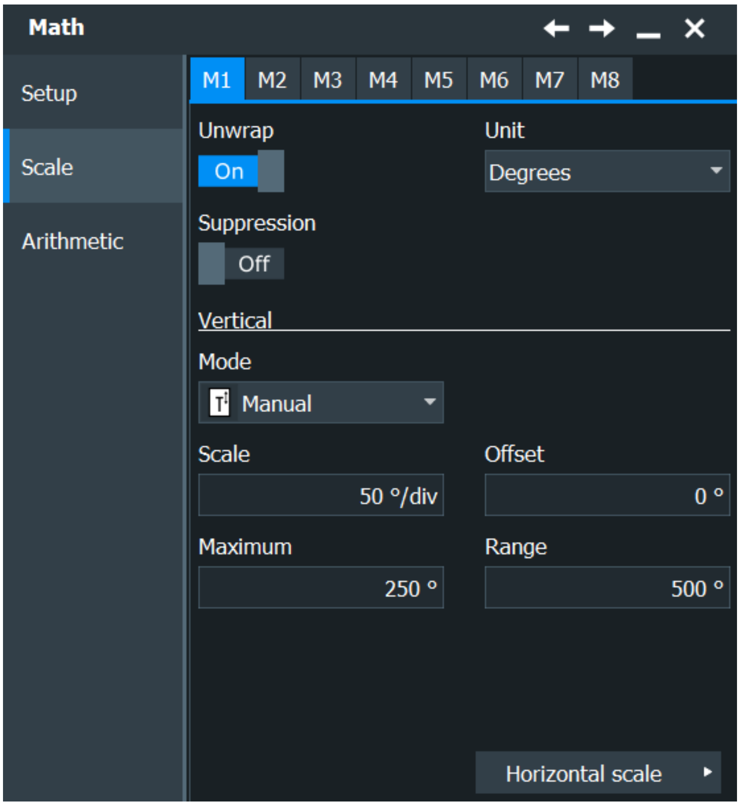
Sets a voltage offset to adjust the vertical position of the math function on the screen. Negative values move the waveform, positive values move it down.

If the "Vertical scaling mode (Manual/Auto)" on page 273 is set to "Auto", this setting is read-only.

Remote command:
`CALCulate:MATH<m>:VERTical:OFFSet` on page 1460

7.3.6.2 FFT scale settings

In this chapter, you can find functions to set the vertical parameters of the FFT math waveform.



See also:

- Vertical scaling mode (Manual/Auto)
- Vertical Offset
- Vertical Scale

Unwrap
If enabled, phase shifts due to a limitation of the value range are eliminated.

Remote command:
`CALCulate:MATH<m>:FFT:PHASe:UNWRap` on page 1575

Unit
Defines the scaling unit for phase display.

- Radians
- Degrees

Remote command:
`CALCulate:MATH<m>:FFT:PHASe:SCALe` on page 1574

Suppression
Enables noise suppression. Phase calculation is restricted to frequencies with a minimum magnitude, the threshold value.

Remote command:
`CALCulate:MATH<m>:FFT:PHASe:SUPPression` on page 1575

Threshold
Defines the minimum frequency magnitude for which phases are calculated. This setting is only available if "Suppression" is enabled.

Remote command:
CALCulate:MATH<m>:FFT:PHASe:THReshold on page 1575

Vertical maximum
Defines the maximum value on y-axis for spectrum displays. Only available for "Manual" scale mode.

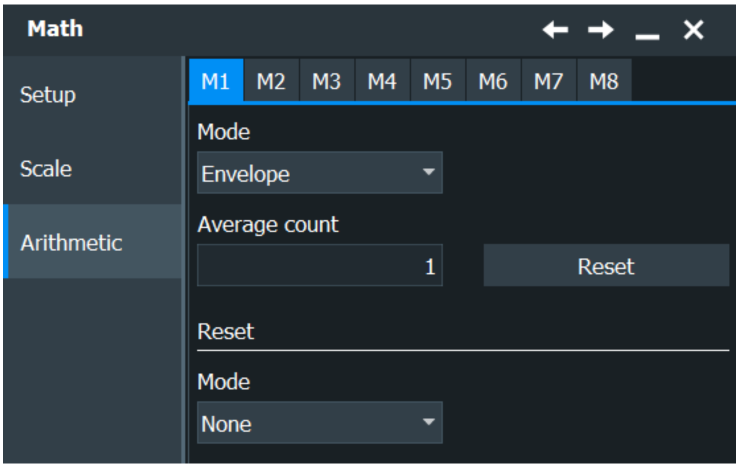
Remote command:
CALCulate:MATH<m>:VERTical:MAXimum on page 1461

Vertical range
Defines the range of FFT values to be displayed.

Remote command:
CALCulate:MATH<m>:VERTical:RANGe on page 1460

7.3.7 Math arithmetic

Access: [Math] > "Arithmetic" tab.



In this tab you can specify the waveform arithmetic for the math waveforms.

Mode
Waveform arithmetic builds the resulting waveform from several consecutive acquisitions and subsequent math calculations of the signal. For details, see "Wfm Arithmetic" on page 136.

"Original"	The original results are displayed.
"Envelope"	The envelope curve of all acquired and calculated results is displayed.
"Average"	The average of all acquired and calculated results is displayed.

"RMS" The root mean square of the math data is displayed. The result is the average power spectrum. If you measure the channel power on this RMS spectrum, you get the same result as for the average channel power measurement on waveforms.

"MinHold" Determines the minimum result for each input value from the data of the current acquisition and some acquisitions before.

"MaxHold" Determines the maximum result for each input value from the data of the current acquisition and some acquisitions before.

Remote command:
`CALCulate:MATH<m>:ARIThmetics` on page 1459

Average count (N-single count)

Access:

- "Menu" > "Acquire" > "Setup" tab > "Average count (N-single count)"
- "Menu" > "Acquire" > "Segmented" tab > disable "Acquire maximum" > "Required"
- [Math] > "Setup" tab > "Mode" is not "Off" > "Average count"

The acquisition and average count has several effects:

- It sets the number of waveforms acquired with [Single]
- It defines the number of waveforms used to calculate the average waveform. Thus, the instrument acquires sufficient waveforms to calculate the correct average if "Average" is enabled for waveform arithmetic. The higher the value is, the better the noise is reduced.
- It sets the number of acquisitions to be acquired in a fast segmentation acquisition series. Thus, you can acquire exactly one fast segmentation acquisition series with [Single].
  If fast segmentation is enabled and configured to acquire the maximum number of acquisitions, the acquisition count is set to that maximum number and cannot be changed.
  See also "Number of acquisitions" on page 138.
- It is the "Finished" criteria for the state of a mask test.

Remote command:
`ACQuire:COUNt` on page 1336

Reset now

Forces the immediate restart of the envelope and average calculation for all waveforms.

Remote command:
`ACQuire:ARESet:IMMediate` on page 1336

Arithmetic Reset > Mode

Defines when the envelope and average evaluation restarts.

"None" No restart, the number of acquisitions considered by the waveform arithmetics is not limited.

"Time" Restarts the envelope and average calculation after the time defined in "Time".

"Waveforms" Restarts the envelope and average calculation after a number of acquired waveforms defined in "Count".

Remote command:
ACQuire:ARESet:MODE on page 1336
ACQuire:ARESet:TIME on page 1336
ACQuire:ARESet:COUNt on page 1337

7.4 History

The history accesses the data of previous acquisitions and provides them for further analysis.

7.4.1 About history

If a continuous acquisition runs, the captured data is stored in the sample memory and the current acquisition is processed and shown on the display. After the acquisition was stopped, the history accesses the captured samples that were stored, displays these samples as history waveforms, and makes them available for further analysis. It considers all channels that were enabled during the running acquisition. When a new acquisition is started with [Run Stop] or [Single], the memory is cleared and written anew.

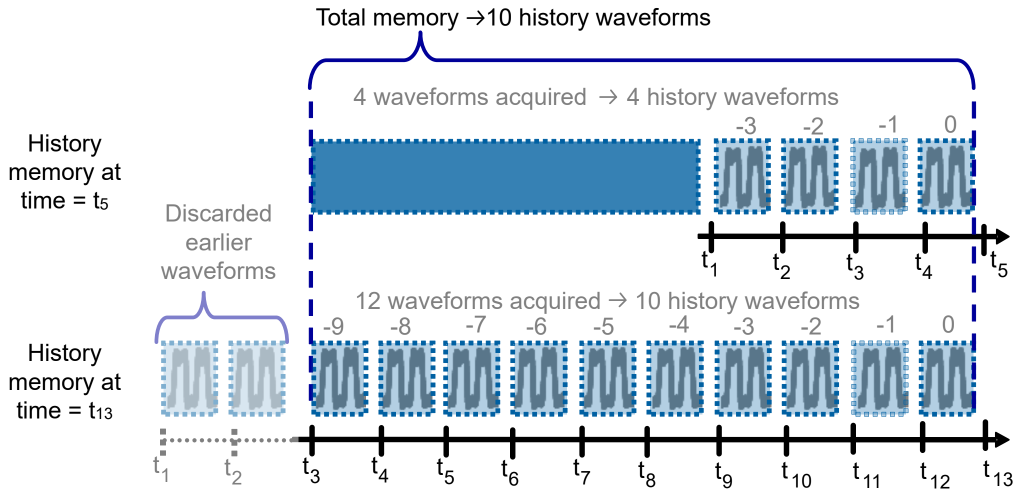


Figure 7-5: History memory. In this example, the memory can store 10 waveforms.

You can work with history waveforms in the same way as with the waveform of the latest acquisition: use zoom, cursor measurements, and automatic measurements, create math waveforms, perform mask testing and so on. Saving the history data is also possible, either completely or a part of the data.

The number of stored history waveforms depends on the memory size, the number of enabled channels, and the record length. The shorter the record length, the less the number of channels, and the larger the memory, the more history waveforms are saved.

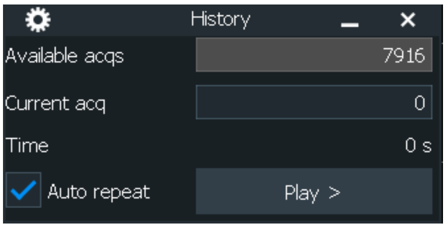
Memory extension

The memory can be enhanced to 1 GSa with option R&S RTO-B110.

Quick-access History dialog box

When you press the [History] key on the front panel or tap "Display" menu > "Show history", the history mode is enabled and the quick-access "History" dialog box is displayed. A running acquistion stops immediately.

The small quick-access "History" dialog box can remain visible on the screen during history replay, so that the history can be replayed at any time by a simple tap on the "Play" button. Closing the quick-access "History" dialog box, or starting a new acquisition disables the history mode.



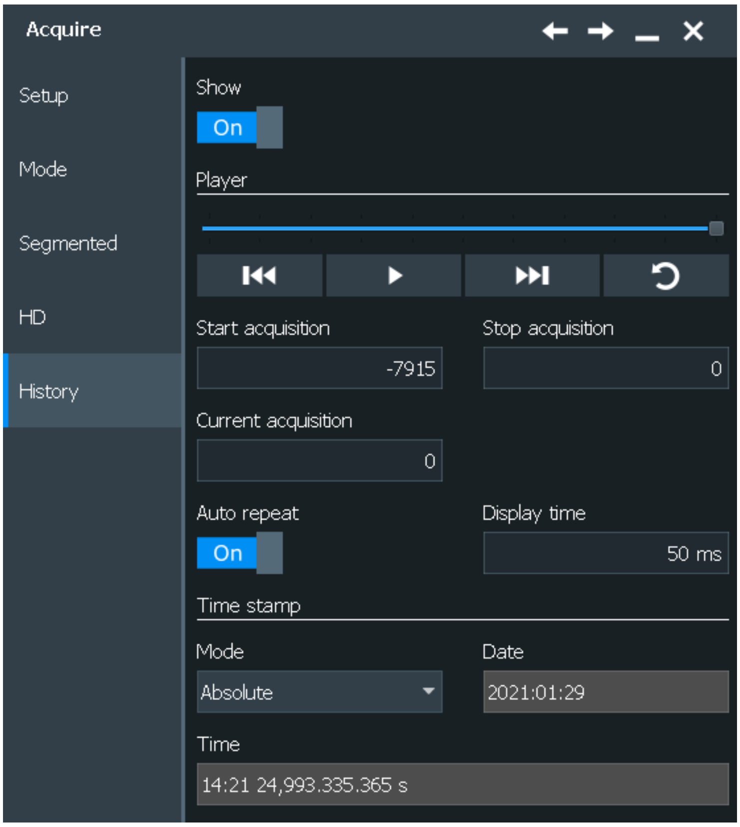
Export of history waveforms

7.4.2 History setup

The "History" dialog contains the complete functionality on history viewing and information.

The most important information and functions are also provided in the quick-access history dialog box.

Access: "Menu" > "Acquire" > "History" tab.



Show history / Export history

Enables the history mode and allows you to save history waveforms to file.

The history display is enabled automatically when you press the [History] button. It is disabled when you close the quick-access "History" dialog box.

For details on data export, see Chapter 12.2.2.2, "Waveform history settings", on page 460.

Remote command:
`CHANnel<m>[:WAVeform<n>]:HIStory[:STATe]` on page 1463

Player

The player can be used to control the playback of the history waveforms.

Sets the oldest acquisition in the sample memory as "Start acquisition" and "Current acquisition".

Starts and stops the replay of the history waveforms from "Start acquisition" to "Stop acquisition".

Sets the newest acquisition in the sample memory as "Stop acquisition" and "Current acquisition". This acquisition always has the index "0".

All acquisitions that are saved in the memory are in the viewer.

Start acquisition
Sets the index of the first (oldest) acquisition to be displayed or exported. The index is always negative.

Remote command:
`CHANnel<m>[:WAVeform<n>]:HISTory:STARt` on page 1464

Stop acquisition
Sets the index of the last (newest) acquisition to be displayed or exported. The newest acquisition of the complete acquisition series always has the index "0".

Remote command:
`CHANnel<m>[:WAVeform<n>]:HISTory:STOP` on page 1464

Current acquisition
Accesses a particular acquisition in the memory to display it, or to save it. The newest acquisition always has the index "0". Older acquisition have a negative index.

If a history replay is running, the field shows the number of the currently shown acquisition.

Remote command:
`CHANnel<m>[:WAVeform<n>]:HISTory:CURRent` on page 1463

Auto repeat
If selected, the replay of the history waveform sequence repeats automatically. Otherwise, the replay stops at the "Stop acquisition".

Remote command:
`CHANnel<m>[:WAVeform<n>]:HISTory:REPLay` on page 1465

Display time
Sets the display time for one acquisition. The shorter the time, the faster the replay is.

The setting takes effect for history replay and the display of a Fast Segmentation series, see Chapter 5.4.3, "Segmented settings", on page 137.

Remote command:
`CHANnel<m>[:WAVeform<n>]:HISTory:TPACq` on page 1465

Time stamp
The time stamp shows the time of the currently displayed history acquisition. Thus, the time relation between acquisitions is always available.

The time stamp "Mode" can be absolute or relative:
- In "Absolute" mode, the instrument shows the date and the daytime of the current acquisition.
- In "Relative" mode, the time difference to the newest acquisition (index = 0) is shown.

The time stamp can be included in waveform data export, see"Time stamps" on page 460.

During history replay, the time value is displayed and updated if the replay speed ("Display time") is slow enough, that is 40 ms or slower.

The quick-access history dialog box always shows the relative time. In the "History Viewer" tab, you can select the time mode.

Remote command:
CHANnel<m>[:WAVeform<n>]:HISTory:TSDate? on page 1466
CHANnel<m>[:WAVeform<n>]:HISTory:TSABsolute? on page 1466
CHANnel<m>[:WAVeform<n>]:HISTory:TSRelative? on page 1466
CHANnel<m>[:WAVeform<n>]:HISTory:TSRReference? on page 1467

7.4.3 Using history

You can access the history waveforms in two ways:

- Display a particular acquisition.
- Replay all or a part of the saved waveforms to track the signal run.

Furthermore, you can export history data to a file.

- "To open the history and get information" on page 282
- "To display a particular acquisition" on page 282
- "To replay history waveforms" on page 283
- "To exit the history" on page 283
- "To save the history data" on page 283

To open the history and get information

1. Press the [History] key on the front panel.

   A running acquisition is stopped, the history mode is enabled and the quick-access "History" dialog box is displayed. The [History] key lights up as long as the history mode is active.

2. Open the full configuration dialog box:
   - Tap the icon.
   - Open "Menu" > "Acquire" dialog > "History" tab.

To display a particular acquisition

1. In the quick-access "History" dialog box, enter the number of the required acquisition in the "Current acq" field. The newest acquisition always has the index "0", older acquisitions have a negative index.

2. Tap "Play" to start.

Alternatively, you can configure and start the history display from the "History" configuration dialog box:

1. Open the "History" configuration tab.

2. If the history mode is off (the [History] key is not illuminated), select "Show".

   The quick-access dialog box is displayed.

3. Drag the slider to the required acquisition. The current number is shown in the "Current acquisition" field.
   Alternatively, enter the number of the required acquisition in the "Current acquisition" field.
4. Tap "Play" to start.

To replay history waveforms

If you want to see the complete acquisition series without any setup, simply tap "Play" in the quick-access "History" dialog box. For specific analysis of history data, use the history "Viewer" setup.

1. Open the "History" configuration dialog tab.
2. If the history mode is off (the [History] key is not illuminated), enable "Show".
   The quick-access dialog box is displayed.
3. Define the part of the history you want to see by doing one of the following:
   - Tap to see the complete history.
   - Enter the "Start acquisition" of the oldest acquisition to display and the "Stop acquisition" of the newest acquisition to display. All waveforms between the two indexes are displayed.
     To enter the oldest or newest acquisition for either index, tap the appropriate button. The newest acquisition always has the index "0". The "Start acquisition" is always negative.
4. Tap to start.

To exit the history

► Choose one of the following ways:
- Close the quick-access "History" dialog box.
- In the "History" configuration tab, disable "Show".
- Start the acquisition.

To save the history data

You can save the complete history, or some subsequent waveforms from the history, or a single history waveform. You can also decide to save the complete waveforms, or a part of each waveform.

1. Open the "Menu" > "Save/Recall" dialog.
2. Select the "Save/Recall" tab.
3. In the "Save " tab, press "Waveform".
4. In the "Setup" tab, tap the source icon to select the waveform you want to save.
5. If you want to save only a part of each waveform, set the "Scope".
   For settings, see "Scope" on page 458.
6. Select the "History" tab.

7. Enable "Export history".
8. If you want to write the timestamps into the data file, enable "Time stamps".
9. To save one waveform out of the history memory:
   a) Make sure that "Multiple acquisitions in one file" is disabled.
   b) Enter the number of the required acquisition in "Acquistion index".
      The newest acquisition in the memory always has the index "0". Older acquisitions have a negative index.
   c) Tap "Save" or "Save As" to save the waveform data to the specified file.
10. To save several subsequent history waveforms:
    a) Enable "Multiple acquisitions in one file".
    b) Define the range of the waveforms to be saved with "Start acquisition" and "Stop acquisition".
    c) Tap "Start export" to play the history and to save the history data to the specified file.

Se also Chapter 12.2.2.2, "Waveform history settings", on page 460.

7.5 XY-diagram

XY-diagrams combine the voltage levels of two waveforms in one diagram. They use the voltage level of a second waveform as the x-axis, rather than a time base. This allows you to perform phase shift measurements, for example. You can display up to four different XY-diagrams.

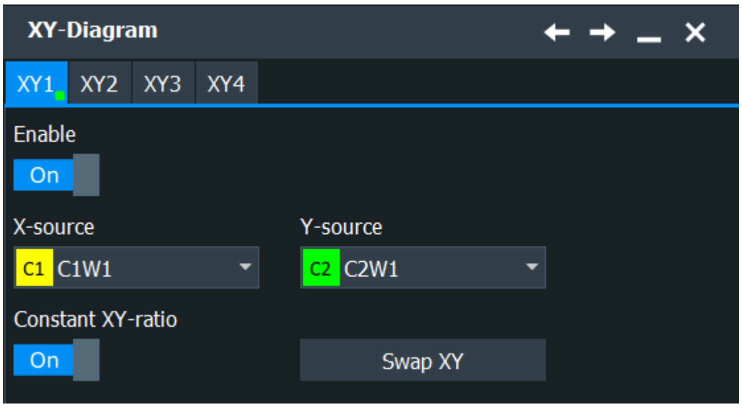
XY-diagrams can be used to display the IQ representation of a signal.

7.5.1 Settings for XY-diagrams

Access: [App Cockpit]>"Analysis" tab > "XY-Diagram"

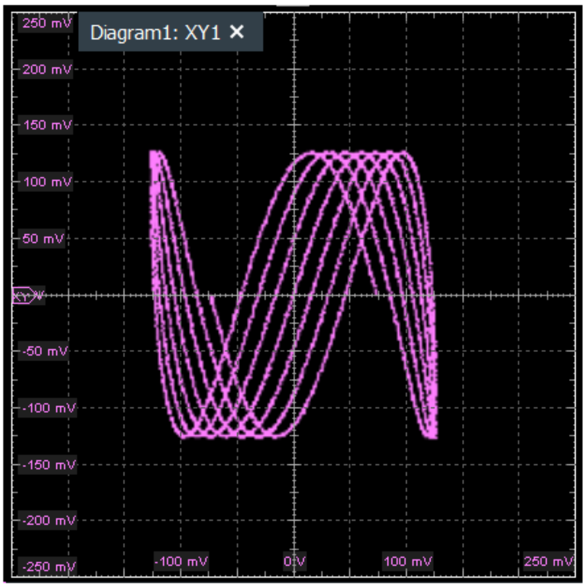
You can display up to four different XY-diagrams that use the voltage level of a waveform as the x-axis, rather than a time base.

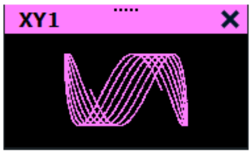
Make sure to select the tab of the required XY-diagram.



Enable

If activated, the XY-waveform is active and shown in a diagram, or it is minimized in a signal icon.





Remote command:

`WAVeform<m>:XYCurve:STATe` on page 1468

X-source

Defines the signal source that supplies the x-values of the XY-diagram. Select one of the following:

- One of the waveforms of any channel
- A reference waveform
- The results of a mathematical function

Remote command:

`WAVeform<m>:XYCurve:XSOurce` on page 1469

Y-source
Defines the source to be used as the y-axis of the XY-diagram. Select one of the following:

- One of the waveforms of any channel
- A reference waveform
- The results of a mathematical function

Remote command:
WAVeform<m>:XYCurve:YSOurce on page 1469

Constant XY-ratio
If enabled, the x- and y-axes maintain a constant ratio in the diagram.

Remote command:
WAVeform<m>:XYCurve:RATio on page 1467

Swap XY
Replaces the source of the x-axis with the source of the y-axis and vice versa.

Remote command:
WAVeform<m>:XYCurve:SWAP on page 1468

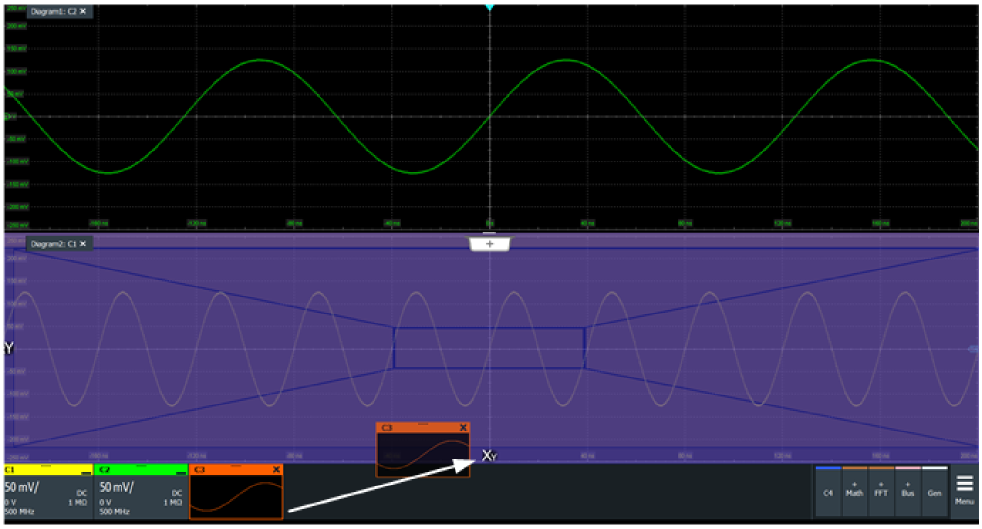
7.5.2 Displaying an XY-diagram

You can create the diagram from active waveforms with drag&drop, or use the dialog box for setup.

To display an XY-diagram with drag&drop

Prerequisites: The source waveform for the y-axis is active in a diagram, the source waveform for the x-axis is either active or minimized.

1. Drag the x-axis waveform to the lower middle of the diagram with the y-axis waveform.
2. Drop the icon when it overlaps with the Xy shown in the middle of the diagram.



The diagram is converted into an XY-diagram.

To set up an XY-diagram

1. Press the [App Cockpit] key.
2. In the >"Analysis" tab, tap the "XY-Diagram".
3. Enable the "XY-diagram".
4. In the "X-source" field, define the signal source that supplies the x-values of the XY-diagram. Select one of the following:
   - One of the waveforms of any channel
   - A reference waveform
   - The results of a mathematical function
5. In the "Y-source" field, define the signal source that supplies the y values of the XY-diagram.
6. To switch the x- and y-values quickly, tap the "Swap XY" button.
7. To maintain a constant ratio while the x- and y-axes are adapted to the acquired data dynamically, activate the "Constant XY-ratio" option.

If the XY-diagram is active or minimized, touch and hold the signal icon to open the "XY-diagram" tab.

8 Measurements

Using the R&S RTO you can perform and display different measurements simultaneously, based on the active signal or math waveforms. The color of the results in the result table corresponds with the source waveform color.

The following measurement methods are available:

- Cursor measurements: measurements can be configured for up to 4 cursor sets to determine specific results at the manually defined cursor positions of an active waveform; the results are displayed in a result box.
- Automatic measurements: up to eight measurements can be configured and performed simultaneously on different sources; the results of each measurement are displayed in a result box.
- Quick measurements: performs a set of automatic measurements on the selected waveform at the push of a button. You can configure the set of measurements.

8.1 Cursor measurements

8.1.1 Cursors and results of cursor measurements

Cursor measurements determine the results at the current cursor positions. The cursors can be positioned manually, or can be configured to follow the waveform. You can measure on one waveform, or on two different waveforms (sources).

Up to 4 cursor sets can be configured and displayed. Each cursor set consists of a pair of horizontal or vertical cursors, or both. Cursor lines can be coupled so that the initially defined distance is always maintained.

The cursors are displayed in the diagrams of the source waveform only, or in all diagrams. For each measurement, labels can be defined for the cursors. By default, the cursors are labeled as C1.1, C1.2, C2.1, C2.2, C3.1, C3.2, C4.1, C4.2.

How to set up cursor measurements is described in Chapter 8.1.2, "Using cursors", on page 289. The Chapter 8.1.3, "Settings for cursor measurements", on page 293 provides a detailed description of all settings.

Cursors can also define a gate to limit the measurement to the section of the waveform between the cursor lines. See Chapter 8.2.3.2, "Gate settings for measurements", on page 309.

The result display of cursor measurements is configurable. Results can be shown in a docked table, or in floating result boxes for each measurement. Similar to waveform diagrams, you can also minimize the result box to a result icon on the signal bar.

For details on using the result box, see Chapter 3.3.7, "Displaying results", on page 63 and "Result position" on page 90.

8.1.1.1 Cursor measurements on time-based waveforms

The cursor for measurement on time-based waveforms returns the following results. The results are displayed automatically when a cursor measurement is enabled.

Figure 8-1: Measurement results of a cursor measurement in time domain

Label	Description
X1, X2	Time at the position of the vertical cursors.
Y1, Y2	Vertical values of the waveform at the position of the horizontal cursors in V or A.
ΔX	Difference between the vertical cursor (time) values
ΔY	Difference between the horizontal cursor values
1/ΔX	Inverse time difference
ΔY/ΔX	Slope of the waveform between the cursors (if measured on one source)

8.1.1.2 Cursor measurements on spectrum waveforms

If the measurement source is a spectrum waveform, the results have a different meaning. Measurement on 2 spectrum waveforms is not possible.

The result box for measurement on spectrum waveforms shows the following information.

Label	Description
X1, X2	Frequency at the position of the vertical cursors
Y1, Y2	Vertical values of the waveform at the position of the horizontal cursors in dB.
Bw	Difference between the vertical cursor (frequency) values
ΔY	Difference between the horizontal cursor values

To set the cursor lines to the peaks, various functions are available in the "Peak Search" tab, see Chapter 8.1.3.3, "Peak search tab", on page 297.

8.1.2 Using cursors

You can start cursor measurements by using the "Cursor" icon on the toolbar, or using the [Cursor] key. For detailed configuration, use the "Cursor" dialog box.

8.1.2.1 Starting a simple cursor measurement

To display cursors using the toolbar

1. Tap the "Cursor" icon on the toolbar.

2. Tap the waveform that you want to measure. Alternatively, you can draw a rectangle in the diagram to position the cursor lines.

   The cursor lines appear and the cursor results are displayed in a table or result box.

To display cursors using the [Cursor] key

1. Select the waveform that you want to measure.
2. Press the [Cursor] key.

   The cursor lines appear and the cursor results are displayed in a table or result box.

To disable one cursor measurement

► If the results are shown in a floating result box, or in a result icon: Close the result box or icon.

► If the results are shown in a docked table:

  a) Tap the "Delete" icon on the toolbar.
  b) Tap a cursor line.

  The cursor set and its measurement results are deleted.

To disable all cursor measurements

► If the results are shown in a docked table:

  a) Tap the "Delete" icon on the toolbar.
  b) Tap the results table.

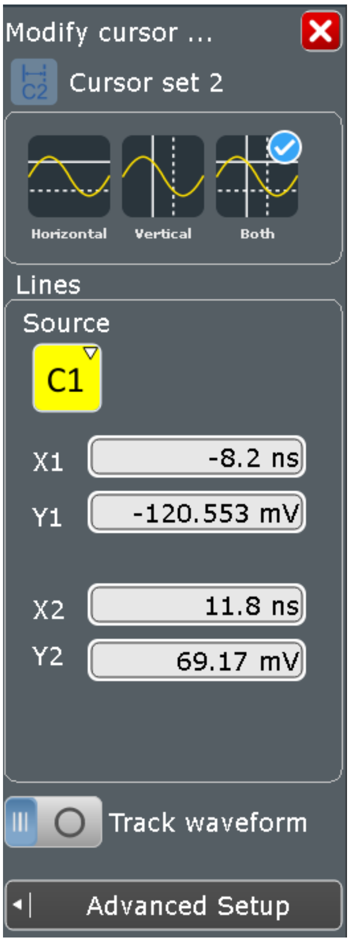
  All cursor sets and measurement results are deleted.

8.1.2.2 Configuring a cursor measurement

To modify the position of the cursor lines, you can drag the lines on the screen. In addition, various settings are possible to refine the measurement.

The complete configuration of cursor measurements is provided in the "Cursors" dialog box.

► You can adjust the basic settings of an active cursor measurement in the sidebar: cursor type, source, positions, and track. To open the sidebar, tap the icon in the result box.



1. To open the "Cursors" dialog box, use one of these ways:
   - Open the "Menu", and select the "Cursor" menu.
   - Press the [Cursor] key.
   - Tap the icon in the result box.
   - On thetoolbar assist, tap "Advanced Setup".
2. Select the "Setup" tab.
3. Select the subtab for the cursor set that you want to use.
4. Tap the source icon, and select the measured waveform. You can select any input channel, or various other active waveforms. Available sources are shown in the source list.
5. If necessary, enable and select a 2nd waveform.

6. Select the cursor type: horizontal, vertical, or both.
7. Define the position of the cursors:
   a) To define exact positions of the cursor lines, enter the X-position for each vertical cursor and the Y-position for each horizontal cursor. If it is not possible to set horizontal cursors, disable "Track waveform".
   b) To position the horizontal cursors automatically, select "Track waveform".
   In this case, cursor 1 indicates the current maximum, cursor 2 indicates the current minimum. If both horizontal and vertical cursors are displayed, the horizontal cursors are placed at the crossing points of the vertical cursors with the waveform. Adjust the vertical cursors manually, and the horizontal cursors follow.
   c) If the waveform arithmetics are set to "Envelope", and "Track waveform" is active, select which horizontal cursor is positioned to the maximum and which to the minimum envelope values.
   d) To keep the distance between the vertical cursors when one cursor is moved, select "Coupling (2 follows 1)".
8. Optionally, select "Show in all diagrams" in the "Advanced" tab. This setting enables the cursor display in all diagrams that are in the same domain as the selected source (time or spectrum).
9. To set the cursors for a spectrum measurement to peak values, select the "Peak Search" tab.
   Tap one of the buttons to place the cursors on the selected peak value. For details, see Chapter 8.1.3.3, "Peak search tab", on page 297.
10. Tap the "Cursor On/Off" icon in the "Setup" tab to activate the cursor measurement.

    The cursors lines and the results are displayed. For details on cursor measurement results, see Chapter 8.1.1, "Cursors and results of cursor measurements", on page 288.

8.1.2.3 Configuring the cursor display

By default, the cursors are displayed as lines in the diagrams and labeled according to the syntax: Cu<cursor set number>.<1|2>

For example, the cursors for the cursor set 2 are labeled 2.1 and 2.2. The horizontal and the vertical cursors lines have the same labels.

You can change the default cursor display and labels.

To set the cursor style

1. Open the "Menu" and select "Settings".
2. Select "Appearance".
3. Select the "Cursor" tab.
4. Select the cursor style. See also: "Cursor style" on page 90.

To add labels to cursor lines

1. Open the "Cursors" dialog box, for example, with "Menu" > "Cursor".
2. Select the subtab for the cursor set you want to configure.
3. Select the "Advanced" tab.
4. For each vertical and horizontal cursor, enter a label.
5. Select "Show labels".

8.1.2.4 Saving measurement results to file

1. Open the "Menu" and select "Save/Recall".
2. Select the "Save" tab.
3. Select "Results".
4. Select the results that you want to export.
5. Save the file.

8.1.3 Settings for cursor measurements

Cursor measurements are configured in the "Cursors" dialog box.

8.1.3.1 Setup tab

The "Setup" tab contains the settings for cursor measurements.

The cursor style and position of the measurement results is defined in the "Settings" > "Appeareance" > "Cursor" dialog box.

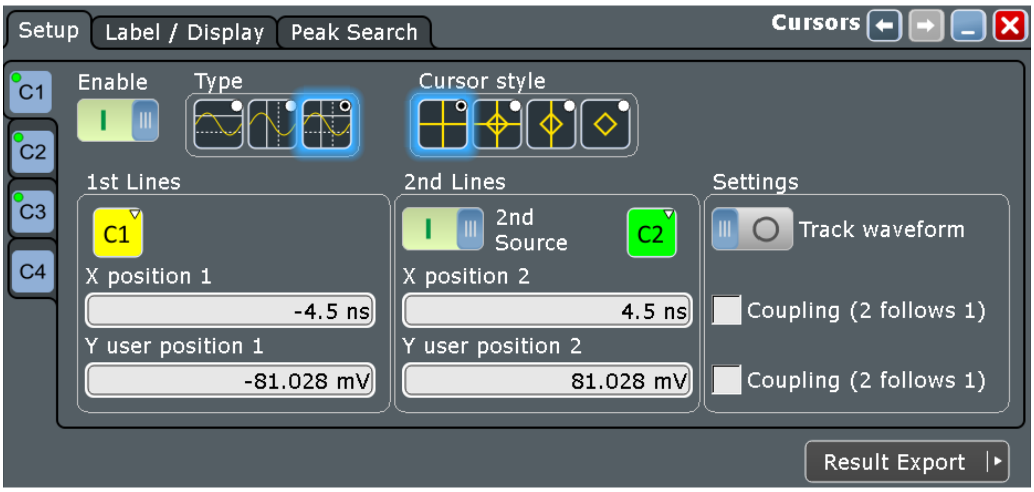


Figure 8-2: Setup for cursor on 2 sources

Cu1, Cu2, Cu3, Cu4
The settings for each cursor measurement (or cursor set) are configured on separate tabs. For each cursor set, a horizontal pair of cursors, a vertical pair of cursors, or both can be displayed.

Cursor On/Off
Enables the selected cursor measurement.

Remote command:
`CURSor<m>:STATe` on page 1470

Type
Defines the cursor type to be used for the measurement.

- Horizontal cursors (Y-cursors)
  Horizontal cursors are positioned automatically along the waveform and can be adjusted manually
- Vertical cursors (X-cursors)
  Both vertical cursor lines are set automatically to the trigger position, and you can reposition them manually.
- Both vertical and horizontal cursors (X&Y-cursors)
  Horizontal cursors are positioned automatically along the waveform and vertical cursors are set to the trigger position. You can reposition all cursor lines manually.

Remote command:
`CURSor<m>:FUNCtion` on page 1471

Source
Defines the source of the cursor measurement. Any of the input signal, math, reference or XY waveforms can be selected.

Remote command:
`CURSor<m>:SOURce` on page 1471

Second source, Source 2
Enables and selects a second source for the cursor measurements. If enabled, the second cursor lines Cx.2 measure on the second source. Using a second source, you can measure differences between two channels with cursors.

Remote command:
`CURSor<m>:USSource` on page 1473
`CURSor<m>:SSOurce` on page 1472

X1 position, X2 position
Defines the position of the vertical cursors.

Remote command:
`CURSor<m>:X1Position` on page 1474
`CURSor<m>:X2Position` on page 1474

Y1 position, Y2 position
Defines the position of the horizontal cursor lines. The setting corresponds to the V1 and V2 values in the "Cursor Results" box.

If "Track waveform" is enabled, the user setting is disabled and the measurement results are displayed in the "Cursor Results" box.

Remote command:
`CURSor<m>:Y1Position` on page 1475
`CURSor<m>:Y2Position` on page 1475

Envelope selection 1|2
Envelope selection is effective under the following conditions:
- The waveform arithmetic of the cursor source waveform is set to envelope waveform (see "Wfm Arithmetic" on page 136)
- "Track waveform" is enabled.
- Both horizontal and vertical cursors are enabled ("Type" = *Both*).

The setting defines which horizontal cursor is positioned to the maximum and which to the minimum envelope values.

"Minimum" The horizontal cursor is set to the crossing point of the vertical cursor with the minimum waveform envelope.

"Maximum" The horizontal cursor is set to the crossing point of the vertical cursor with the maximum waveform envelope.

Remote command:
`CURSor<m>:X1ENvelope` on page 1476
`CURSor<m>:X2ENvelope` on page 1477

Track waveform
The horizontal cursors track the waveform. The first cursor line indicates the current vertical minimum, and the second cursor line indicates the maximum. If the waveform changes, e.g. during a running measurement, the cursors move along with it. If both horizontal and vertical cursors are displayed, the horizontal cursors are positioned to the crossing points of the vertical cursors with the waveform. The measurement results are displayed in the "Cursor Results" box.

Tracking disables the Y-coupling (coupling horizontal cursor lines) and the Y user position settings.

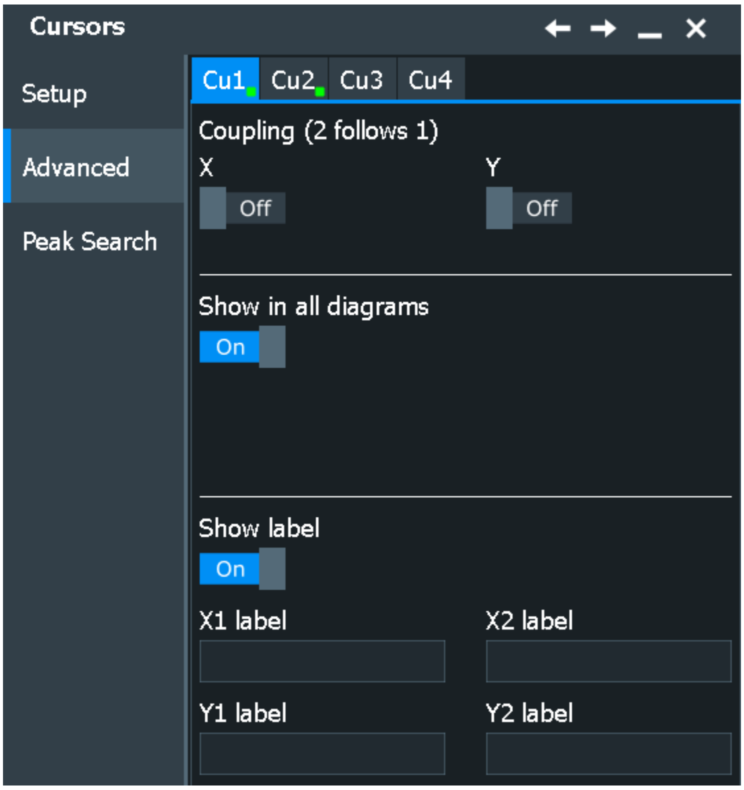
Remote command:
`CURSor<m>:TRACking[:STATe]` on page 1474

8.1.3.2 Advanced settings

Access: Cursor key > "Advanced" tab

The settings in the "Advanced" cursor tab configure the behavior and display of cursor lines, and labels for the lines.

The cursor style and position of the measurement results is defined in the "Settings" > "Appeareance" > "Cursor" dialog box.See Chapter 4.3.7, "Cursor appearance settings", on page 89.



Cu1, Cu2, Cu3, Cu4

The settings for each cursor measurement (or cursor set) are configured on separate tabs. For each cursor set, a horizontal pair of cursors, a vertical pair of cursors, or both can be displayed.

Coupling (2 follows 1)

Couple the horizontal and vertical cursor lines so that the distance between the two lines remains the same if one cursor is moved.

Remote command:
`CURSor<m>:YCOupling` on page 1476
`CURSor<m>:XCOupling` on page 1475

Show in all diagrams

Shows the enabled cursor measurements in all active diagrams of the same (time/ spectrum) domain.

Remote command:
`CURSor<m>:SIAD` on page 1477

Show label

Shows the cursor labels in the diagram.

Remote command:
`CURSor<m>:LABel` on page 1477

X1 label, X2 label
Defines a label to be displayed with the vertical cursors.

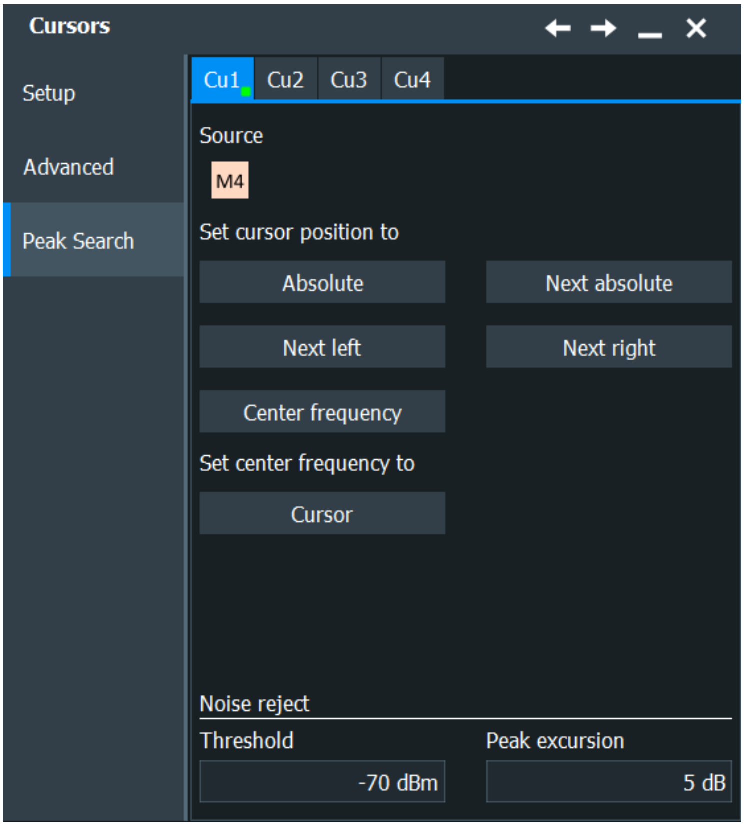
By default, the cursors are labeled as C1.1, C1.2, C2.1, C2.2, C3.1, C3.2, C4.1, C4.2.

Y1 label, Y2 label
Defines a label to be displayed with the horizontal cursors.

8.1.3.3 Peak search tab

Access: [Cursor] key > "Peak Search" tab

The settings on this tab are only available in spectrum mode, i.e. the source of the cursor measurement is an FFT math waveform. In this case, the cursors can indicate the results of a peak search on the waveform. You can define which peaks the instrument determines by defining the noise reject settings.



Cu1, Cu2, Cu3, Cu4
The settings for each cursor measurement (or cursor set) are configured on separate tabs. For each cursor set, a horizontal pair of cursors, a vertical pair of cursors, or both can be displayed.

Source
Defines the source of the cursor measurement. Any of the input signal, math, reference or XY waveforms can be selected.

Remote command:
CURSor<m>:SOURce on page 1471

Absolute
Both cursors are set to the absolute peak value.

Remote command:
CURSor<m>:MAXimum[:PEAK] on page 1479

Next absolute
Cursor 2 is set to the next smaller absolute peak from the current position.

Remote command:
CURSor<m>:MAXimum:NEXT on page 1480

Next left
Cursor 2 is set to the next peak to the left of the current position.

Remote command:
CURSor<m>:MAXimum:LEFT on page 1480

Next right
Cursor 2 is set to the next peak to the right of the current position.

Remote command:
CURSor<m>:MAXimum:RIGHt on page 1480

Center frequency
Sets the vertical cursor line Cu1 to the center frequency.

Remote command:
CURSor<m>:FFT:TOCenter on page 1479

Set center frequency to Cursor
Sets the center frequency to the frequency value that is measured at cursor line Cu1.

Remote command:
CURSor<m>:FFT:SETCenter on page 1479

Threshold
Defines an absolute threshold as an additional condition for the peak search. Only peaks that exceed the threshold are detected.

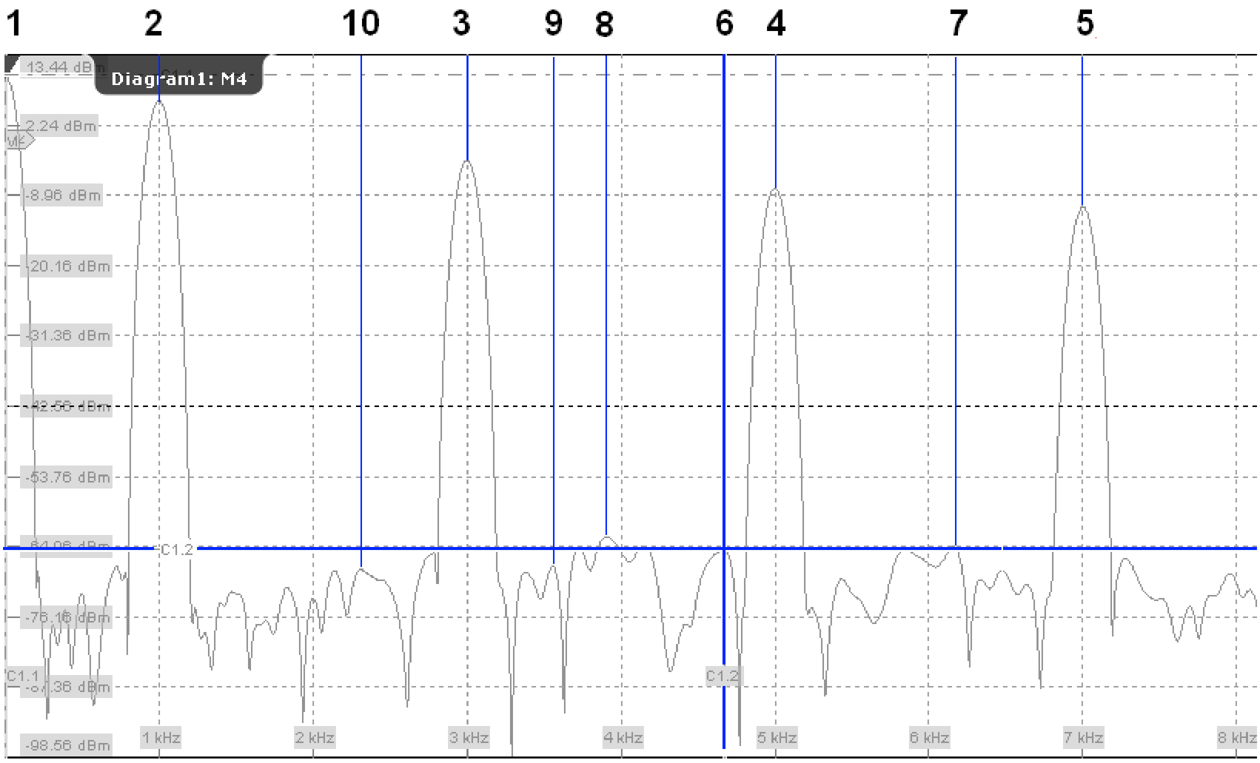
This setting is only available for spectrum waveforms. It is valid for cursor measurements, spectrum measurements and peak search.

Remote command:
CURSor<m>:THReshold on page 1629
MEASurement<m>:SPECtrum:ATHReshold on page 1515

Peak excursion
Defines a relative threshold, the minimum level value by which the waveform must rise or fall to be considered as a peak. To avoid identifying noise peaks, enter a peak excursion value that is higher than the noise levels.

This setting is only available for spectrum waveforms. It is valid for cursor measurements, spectrum measurements and peak search.

The following figure shows a cursor measurement on a spectrum waveform:



If "Peak excursion" is 30 dB, the peaks 1 to 5 are found. If "Peak excursion" is 20 dB, also the peaks 6 to 10 are found. The cursor position is on peak 6.

Remote command:
`CURSor<m>:PEXCursion` on page 1628
`MEASurement<m>:SPECtrum:PEXCursion` on page 1515

8.2 Automatic measurements

The R&S RTO can perform many measurements in parallel. The measurements are combined in up to 8 measurement groups. In addition, you can run the quick measurement.

The basic measurement settings are source, category, and the selection of the measurement. You can refine the setup to get more specific results:

- **Gating**
  A gate limits the measurement to a user-defined part of the waveform. See Chapter 8.2.3, "Measurement gates", on page 308.
- **Statistics and long term measurements**
  To evaluate time-dependent behavior of measurement results, you can use statistics, long term measurements, and tracks. You can also decide, how many mea-

surement results per acquisition contribute to the calculation. See: Chapter 8.2.11, "Result analysis", on page 356.

- **Limit checks and actions on test result**
  Limit and margin checks evaluate if the measurement result exceeds a specified value. You can define actions that are performed on limit or margin violation. See: Chapter 8.2.12, "Limit and margin checks", on page 365.

Measurement categories

The various measurement are grouped in several categories. The category defines which sources can be analyzed.

Time domain:

- Amp/Time: amplitude and time measurements
- Eye: eye diagram measurements
- Histogram: measurements on histograms
- Jitter measurements (available if one of the jitter options is installed)
- Protocol: available for audio signals (option R&S RTO-K5)
  Protocol: advanced analysis (available if option R&S RTO-K35 is installed)

Frequency domain

- Spectrum: measurements on spectrum waveforms
- Histogram: measurements on histograms

Details on automatic measurements are described in the following chapters:

8.2.1 Measurement setup in general

Up to 8 measurement groups can be defined. Each measurement group is configured in its own subtab. For each measurement group, the category and the source are defined in the "Meas Group" tab, and you can also enable statistic evaluation. Available sources and measurements depend on the selected category, for example, histogram measurements need a defined histogram as source. For each measurement group, you select all required measurements. If further settings are available for a measurement, a settings icon is shown beside the measurement's name.

8.2.1.1 Starting an automatic measurement

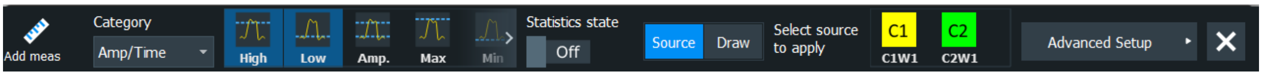
There are three methods to start an automatic measurement, each with slightly different effects:

- Using the "Measure" icon on the toolbar:
  See: "To start a measurement using the toolbar icon" on page 301.
- Pressing the [Meas] key on the front panel.
  See: "To start a measurement with the [Meas] key" on page 301.
- See: Chapter 8.2.1.2, "Configuring measurements", on page 301.

To start a measurement using the toolbar icon

1. Tap the "Measurement" icon on the toolbar.

2. On the toolbar assist, select the category, the measurements and the source to be measured.



3. Define the measurement range in one of these ways:
   - To measure the complete waveform, tap the diagram with the waveform.
   - To define a gate that limits the measurement, draw a rectangle on the screen.

   The "Meas Results" are displayed.

To start a measurement with the [Meas] key

1. Select the waveform on the screen.
2. Press the [Meas] key.

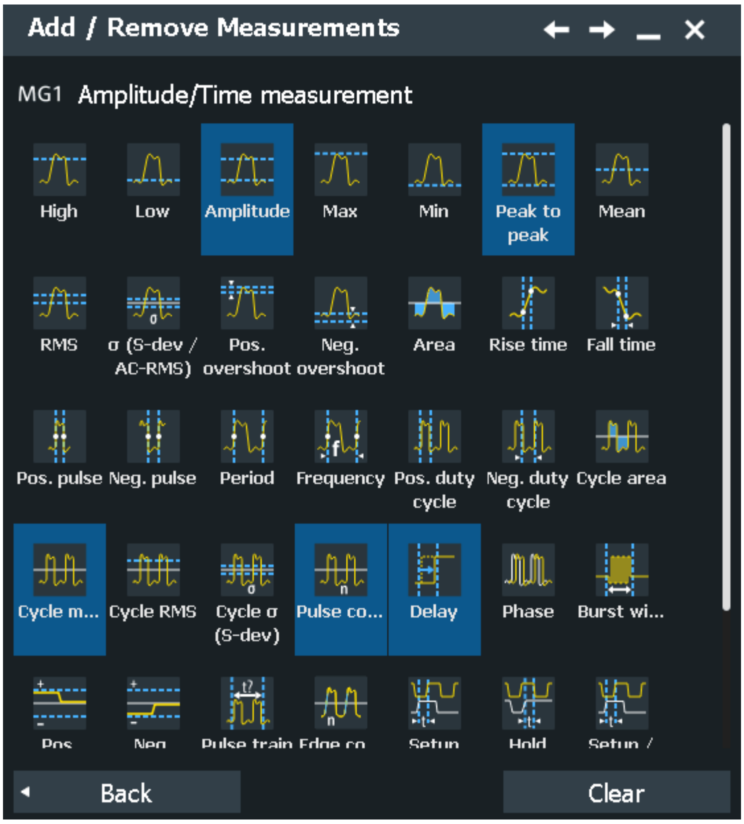
   The measurement for the selected waveform is enabled using the next available measurement configuration. The "Meas Results" are displayed.

8.2.1.2 Configuring measurements

1. To open the "Measurements" dialog box, choose one of these ways:
   - If a measurement is already running, tap the result table. Then press the [Meas] key.
   - If no measurement is running, open the menu and select "Measure".

   If you double-tap the title of a result table, the toolbar assist opens. Here you can change the measurement types and enable statistics. If you tap "Advanced Setup" in the toolbar assist, the "Measurements" dialog box opens.
2. Select the subtab for the measurement group that you want to configure.
3. Select the measurement "Category", for example, "Amp/Time".

4. Tap "Source" and select the waveform to be measured.
   Spectrum measurements require an FFT math waveform as measurement source. Histogram measurements require a histogram as source.
5. Tap "Add / Remove" and select all measurements that you want to include in the measurement group. Tap "Clear" to deselect all settings.



6. Tap "Back" to close the selection box.
   All selected measurements are displayed in the list of active measurements.
7. A "Settings" icon indicates whether further settings are required. Tap the measurement. A dialog box opens, and you can adjust the measurement.
   The settings are explained in the following chapters:
   - Chapter 8.2.5.2, "Settings for amplitude/time measurements", on page 326
   - Chapter 8.2.6.2, "Settings for eye diagram measurements", on page 337
   - Chapter 8.2.7.2, "Settings for spectrum measurements", on page 339
   - Chapter 8.2.8.5, "Settings for histogram measurement", on page 349
8. Optionally, define a gate to restrict the measurement to a part of the waveform. See Chapter 8.2.3.1, "Using measurement gates", on page 308.
   If you enabled the measurement with the toolbar icon and drew a rectangle on the diagram, the gate is already defined and enabled.
9. To compile and display statistics for the measurement, enable "Statistics". See also Chapter 8.2.11.1, "Statistics", on page 357.

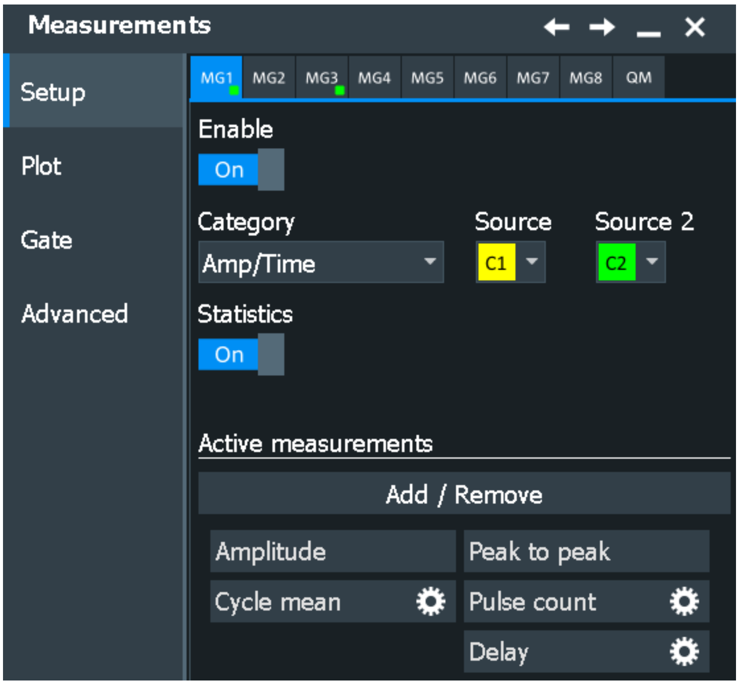
10. Optionally, perform a limit or margin check as described in Chapter 8.2.12.1, "Performing limit checks", on page 365.
11. "Enable" the measurement group.

    The measurement results are displayed by default below the waveform diagram.

8.2.1.3 General measurement settings

Automatic measurements are configured in the "Measurements" dialog box.

Access: [Meas] > "Meas Group" tab



This section describes the settings that relate to all measurements. Specific settings are described in the corresponding category chapters:

- Chapter 8.2.5.2, "Settings for amplitude/time measurements", on page 326
- Chapter 18.1, "Jitter measurements (Options R&S RTO-K12/-K13)", on page 1066
- Chapter 8.2.6.2, "Settings for eye diagram measurements", on page 337
- Chapter 8.2.7.2, "Settings for spectrum measurements", on page 339
- Chapter 8.2.8.5, "Settings for histogram measurement", on page 349

MG 1/2/3/4/5/6/7/8/Quick Meas
Subtabs for each measurement group, and for the quick measurement. The subtabs contain the configuration settings. A green dot on the tab indicates that the measurement group is active.

Enable
Starts the measurements of the selected measurement group.

Remote command:
MEASurement<m>[:ENABle] on page 1481

Category
Measurement category. The following categories are available:

Time domain:
- Amp/Time: amplitude and time measurements
- Eye: eye diagram measurements
- Histogram: measurements on histograms
- Jitter measurements (available if one of the jitter options is installed)
- Protocol: available for audio signals (option R&S RTO-K5)
  Protocol: advanced analysis (available if option R&S RTO-K35 is installed)

Frequency domain
- Spectrum: measurements on spectrum waveforms
- Histogram: measurements on histograms

Remote command:
MEASurement<m>:CATegory on page 1484

Source, Source 2
Define the source of the measurement group. The 2nd source is required for amplitude/time measurements that are performed on two waveforms (e.g. delay, phase). Availability of sources depends on the selected category and installed options.

Remote command:
MEASurement<m>:SOURce on page 1482

Active measurements, Add / Remove
The "Active Measurements" list shows the measurements that are selected for the current category. At least, one measurement must be selected. If further settings are available for a measurement, a settings icon is shown beside the measurement's name.

To change the selection, tap "Add / Remove Measurements".

In the "Add / Remove Measurements" dialog box, select the measurements that you need. Tap "Clear" to disable all selected measurements. Tap "OK" to confirm the selection.

Remote command:
MEASurement<m>:MAIN on page 1485
MEASurement<m>:ADDitional on page 1486

Envelope
This setting is only available for measurements on envelope waveforms, see "Wfm Arithmetic" on page 136.

"Both"	The upper and the lower envelope are used in measurements. For time measurements, the averages of min and max values are used, that is, the measurement is performed on the average waveform built from the upper and lower envelope.
"Maximum"	Measurements are performed on the upper envelope.
"Minimum"	Measurements are performed on the lower envelope.

Remote command:
MEASurement<m>:ENVSelect on page 1494

Statistics

Enables the calculation and display of statistical results.

Remote command:
MEASurement<m>:STATistics[:ENABle] on page 1531

8.2.2 Measurement results

By default, the results of automatic measurements are displayed below the waveform diagram when an automatic measurement is enabled.

Meas Group 1 C1		Meas Group 2 C2	
High	122.53 mV	Rise time	11.847 ns
Low	-124.51 mV	Fall time	---
Amplitude	247.04 mV	Pos. pulse	20.05 ns
Frequency	25 MHz	Neg. pulse	20 ns
Cycle RMS	87.999 mV		

If you want to save space in the display, drag the results to the signal bar. The most important results are displayed and updated in a results icon.

The function "Clear all" in the "Display" menu resets all results including long-term measurement and statistic results, and also deletes all waveforms and the history.

Which results are displayed depends on the selected measurements and is described in detail in the following chapters.

The following additional results are available:

- **Statistics**
  You can enable statistical evaluation of the measurement results, and select the statistical results that you want to see. Statistic information is provided in the result box. Stopping and restarting the acquisition does not reset statistics but only stops and continues them.
  See Chapter 8.2.11, "Result analysis", on page 356
- **Measurement histograms**
  The results of measurements can be displayed in a histogram which shows the density distribution of the measurement results in a graphic and thus illustrates the statistics of the measurements.
  See Chapter 8.2.8, "Histograms and histogram measurements", on page 343
- **Long-term measurements**
  Long-term measurements show the behavior of measurement results over a longer time or for many samples. You can define the number of long-term points and export the long-term data, including statistical results. The measurement histogram is a vertical histogram shown in the long-term diagram.
  Long-term measurements are performed on the measurement that is selected for analysis and math on the "Result Analysis" tab.
  See: Chapter 8.2.11, "Result analysis", on page 356

- **Intermediate results**
  You can display auxiliary result lines and reference levels in the source diagram, see Chapter 8.2.2.2, "Configuring the results display", on page 307.

Remote commands:
- `MEASurement<m>:ARES?` on page 1487
- `MEASurement<m>:ARNames` on page 1488
- `MEASurement<m>:RESult[:ACTual]?` on page 1488
- `MEASurement<m>:RESult:COUNt?` on page 1490

8.2.2.1 Measurement status

The overall status of measurement results is indicated by various icons. In general, a question mark before the result value indicates that the measurement result might not be correct due to insufficient amplitude level. Check your amplitude and reference level settings. The icon colors indicate the state of the limit and margin checks.

Icon	Description
No icon, no result value ("---")	The instrument cannot measure the required value, for example, if the acquisition does not contain at least one complete period for frequency and cycle measurements. Check and adjust the waveform settings to get results.
	The measurement result might not be correct due to insufficient amplitude level. Check your amplitude and reference level settings. Limit and margin checks are disabled.
	Limit and margin checks passed, measurement results are reliable.
	The measurement result might not be correct due to insufficient amplitude level. Check your amplitude and reference level settings. Limit and margin checks passed.
	The measurement result might not be correct due to insufficient amplitude level. Check your amplitude and reference level settings. Margin checks failed.
	Margin checks failed.
	The measurement result might not be correct due to insufficient amplitude level. Check your amplitude and reference level settings. Limit checks failed.
	Limit checks failed.

8.2.2.2 Configuring the results display

The measurement results can be displayed in a table below the waveform diagrams, in a floating result box, or in a minimized result icon on the signal bar.

The display settings for measurements are provided on the "Gate/Display" tab, see Chapter 8.2.2.3, "Display settings for results", on page 307.

To clear the measurement results

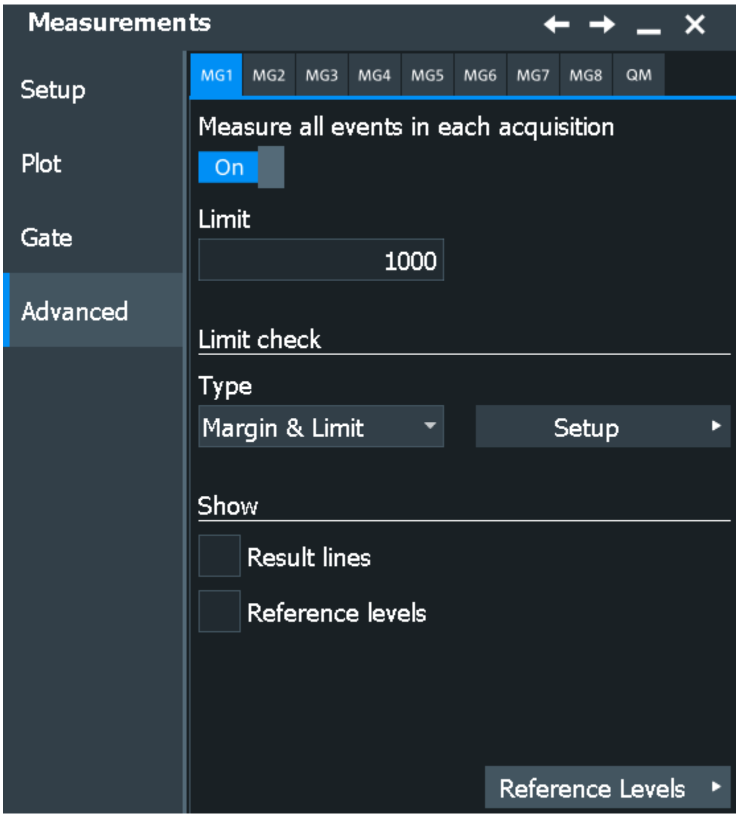
1. On the "Display" menu, tap "Clear all".
2. To restart measurement statistics, without deleting other results, select "Reset" on the "Result Analysis" tab.

   The results in the selected measurement result box are cleared and written anew.

8.2.2.3 Display settings for results

Display settings are specific for each measurement group.

Access: [Meas] > "Advanced" tab



General display setting for measurements are provided in "Settings" > "Appearance" > "Measurements", see Chapter 4.3.8, "Measurement appearance settings", on page 90.

Result lines

Result lines mark the samples in the waveform that are used to obtain the measurement result. For example, maximum, minimum, high and low values, and mean are merked by horizontal lines; start and end are marked by vertical lines.

Remote command:
`MEASurement<m>:DISPlay:RESults` on page 1530

Reference levels

Displays the reference levels used for the measurement in the diagram.

Remote command:
`MEASurement<m>:DISPlay:LEVels` on page 1529

Result control panel

Extends the result box of the selected measurement group with the source settings and the statistics enabling. Thus you can check and change the measurement sources directly in the results box, and also enable statistics there. The setting is only available if the result boxes are not grouped ("Group result dialogs" = Off).

8.2.3 Measurement gates

Gate areas limit the measurement to a user-defined range of the waveform. The gate settings are defined on the "Gate " tab.

Each measurement group can use its own gate. Make sure to select the correct measurement group.

8.2.3.1 Using measurement gates

If you have enabled the measurement with the toolbar icon and drew a rectangle on the diagram, the gate is automatically defined and enabled. If you want to create a gate for a running measurement, or if you want to modify the gate area, configuration is done in the "Measurements" > "Gate" dialog box.

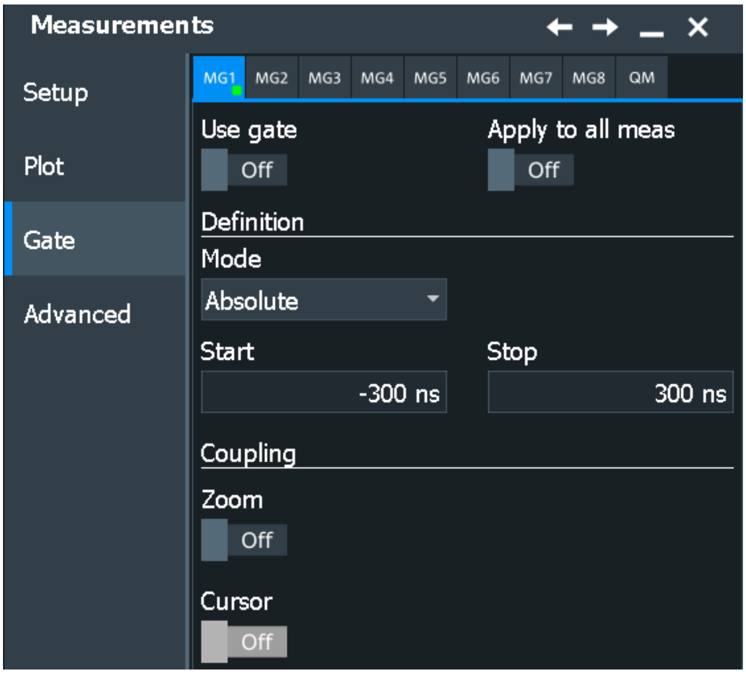
1. In the "Measurements" dialog, select the "Gate" tab.
2. Select the subtab for the measurement group you want to configure.
3. To define the gate, use one of the following methods:
   - Define the start value and the stop value of the gate area by entering either absolute or relative values.
   - If a zoom area has already been defined for the waveform, couple the gate area to the zoom area by selecting the "Zoom coupling" option.
   - If a cursor measurement has already been defined for the waveform, couple the gate area to the cursor lines by selecting the "Cursor coupling" option.
4. If you want to use the same gate for all measurement group, enable "Apply to all meas".

5. Tap the "Use gate" icon to enable the gate usage.

   The measurement is performed on the selected part of the waveform. The gate is shown in the diagram.

8.2.3.2 Gate settings for measurements

Access: [Meas] > "Gate" tab



Result display settings are described in Chapter 8.2.2.3, "Display settings for results", on page 307.

Use gate
Considers the gating settings for the selected measurement and displays the gate.

Remote command:
`MEASurement<m>:GATE[:STATe]` on page 1544

Apply to all meas
If you enable the gate coupling, the gate settings of the selected measurement are copied to all other measurements. Thus, all measurements use the same gate. If zoom or cursor coupling is active in a measurement, the zoom size and cursor positions are adjusted.

Remote command:
`MEASurement<m>:GATE:GCOupling` on page 1547

Gate description
Defines whether the gate settings are configured using absolute or relative values.

"Absolute" The gate is defined by absolute start and stop values.

"Relative" The gate's start and stop values are defined by a percentage of the value range.

Remote command:
CALCulate:MATH<m>:FFT:GATE:MODE on page 1571
MEASurement<m>:GATE:MODE on page 1545
SEARch:GATE:MODE on page 1629

(Relative) Start
Defines the starting value for the gate.

Remote command:
CALCulate:MATH<m>:FFT:GATE:ABSolute:STARt on page 1571
CALCulate:MATH<m>:FFT:GATE:RELative:STARt on page 1572
MEASurement<m>:GATE:ABSolute:STARt on page 1545
MEASurement<m>:GATE:RELative:STARt on page 1545
SEARch:GATE:ABSolute:STARt on page 1630
SEARch:GATE:RELative:STARt on page 1630

(Relative) Stop
Defines the end value for the gate.

Remote command:
CALCulate:MATH<m>:FFT:GATE:ABSolute:STOP on page 1571
CALCulate:MATH<m>:FFT:GATE:RELative:STOP on page 1572
MEASurement<m>:GATE:ABSolute:STOP on page 1545
MEASurement<m>:GATE:RELative:STOP on page 1545
SEARch:GATE:ABSolute:STOP on page 1630
SEARch:GATE:RELative:STOP on page 1631

Zoom
Zoom coupling is available if a zoom is defined. As long as "Zoom coupling" is enabled, the gate area is defined identically to the zoom area - if you change the zoom, the gate changes as well.

If several zoom diagrams are defined, select the zoom diagram to be used for gating. The "Start" and "Stop" values of the gate are adjusted accordingly.

Zoom coupling can be set for measurement gates, FFT gates, and search gates. The zoom must be defined on the diagram that contains the signal source of the measurement, FFT, or search.

Remote command:
MEASurement<m>:GATE:ZCOupling on page 1546
MEASurement<m>:GATE:ZDIagram on page 1547
CALCulate:MATH<m>:FFT:GATE:ZCOupling on page 1572
SEARch:GATE:ZCOupling on page 1631
SEARch:GATE:ZDIagram on page 1631

Cursor
If enabled, the gate area is defined by the cursor lines of an active cursor measurement.

If several cursor measurements are enabled, select the cursor set to be used for gating. The "Start" and "Stop" values of the gate are adjusted to the values of the cursor line positions. The measurement is limited to the part of the waveform between the cursor lines.

Remote command:
`MEASurement<m>:GATE:CCOupling` on page 1546
`MEASurement<m>:GATE:CURSor` on page 1546

8.2.4 Reference levels

Some measurement require reference levels to obtain the measurement points, e.g. time measurements or pulse count. Reference levels are referred to the signals, for each waveform you can define specific reference levels. Thus, for all measurements on a waveform the same reference levels are used.

Usually, reference levels are determined automatically. The instrument determines the high and low signal levels based on amplitude and histogram measurements of the acquisition. The reference levels are set relatively to the determined signal levels.

However, for irregular data and in special measurement setups it may be useful to configure the levels manually:

- Data signals can contain intervals where no data is transmitted, so that a high and low state cannot be determined for each acquisition. In this case, you can define the high and low signal levels manually to evaluate other measurement results.
- If the signal levels vary strongly or have large overshoots, the rise and fall levels may be difficult to determine.
- If fixed levels are defined for the DUT, you can configure the reference levels in the R&S RTO correspondingly and analyze the resulting measurement data.

In manual configuration, the reference levels can be set relatively to defined signal levels or as absolute values. You can also set the reference levels directly.

In addition to reference and signal levels, you can define a hysteresis for the middle reference level and tubes for signal levels. Hysteresis is useful for measurements that determine zero-crossings. Period, frequency, and pulse measurements are based on hysteresis - the instrument returns results if the amplitude of the signal exceeds the hysteresis. Thus, measurement during the transient oscillation is also possible. Tubes define evaluation ranges for measurements that require detection of the high level or low level. If the signal value remains within the defined tubes, it is considered to be high or low.

Reference levels and result lines can be displayed in the diagram, see "To display reference levels and result lines" on page 314.

8.2.4.1 Configuring reference levels

To determine reference and signal levels automatically

By default, the histogram of the measurement data is evaluated to determine the required levels automatically. However, you can define several parameters to adapt the evaluation to your data.

1. Open "Menu" > "Measure" > "Advanced" tab > "Reference Level" > "Levels" tab.
2. Define the "Source", the waveform for which the reference is defined. The source can be any signal input, math or reference waveform.
3. Select "Mode" = "Auto" .
4. Define the "Signal level", the method which is used to determine the signal levels. For details, see "Signal level" on page 316.
5. By default, the lower reference level is defined at 10% of the signal amplitude, the middle reference level at 50% and the upper reference level at 90%. You can select other "Relative levels" to be used for evaluation.
   If default percentages do not fit, select "User defined" and enter the percentages for the upper, middle, and lower reference levels.
   The signal levels are determined by the instrument.
6. To determine the reference levels using average values from several histograms, enable the "Histogram averaging" option and define an "Average Count" to define how many histograms are averaged.
   Averaging is not available if "Absolute peak" are selected as the "Signal level".

To determine reference levels manually

You can configure the reference levels manually as fixed absolute or relative values.

1. Open "Menu" > "Measure" > "Advanced" tab > "Reference Level" > "Levels" tab.
2. Define the "Source", the waveform for which the reference levels are defined. The source can be any signal input, math or reference waveform.
3. Select "Mode" = "Manual" .

4. Under "Level definition", select whether you want to define the levels using absolute or relative values.
5. Under "User level", select whether you want to configure the high and low signal levels ("Signal") or the lower, middle and upper reference levels ("Reference").
6. To define high and low signal levels if "Signal" is selected:
   a) Enter the absolute high and low signal levels.
   b) If "Level definition" is relative, select one of the predefined "Relative levels". If default percentages do not fit, select "User defined" and enter the percentages for the upper, middle, and lower reference levels.
      The upper and lower reference levels are computed from the signal level values and the percentage values.
   c) If "Level definition" is absolute, set the absolute "Top distance" and "Bottom distance" values, the differences between signal and reference levels.
      The upper and lower reference levels are computed from the signal level values and the distances.
7. To define lower, middle and upper reference levels if "User reference level" is selected:
   a) Enter the absolute upper and lower reference levels.
   b) If "Level definition" is relative, select one of the predefined "Relative levels". If default percentages do not fit, select "User defined" and enter the percentages for the upper, middle, and lower reference levels.
      The high and low signal levels are computed from the reference level values and the percentage values.
   c) If "Level definition" is absolute, set the absolute "Top distance" and "Bottom distance" values, the differences between signal and reference levels.
      The high and low signal levels are computed from the reference level values and the distances.

To define hysteresis and tubes

1. To define a hysteresis for the middle reference level:
   a) Select the "Advanced" tab.
   b) Enter a percentage of the selected signal level.

   A rise or fall from the middle reference value that does not exceed the hysteresis is rejected and not considered a zero-crossing.
2. To define a tube for the high and low signal levels:
   a) In the "Relative outer (RO)" field, define a percentage of the signal level by which the absolute signal level may be larger than high signal level or lower than the low signal level.
   b) In the "Relative inner (RI)" field, define a percentage of the signal level by which the absolute signal level may be higher than the low signal level or lower than the high signal level.

To display reference levels and result lines

1. Open "Menu" > "Measure" > "Advanced" tab.
2. Select the tab for the measurement you want to configure.
3. Enable "Result lines" or "Reference levels" option, or both.

   The reference levels and intermediate results are displayed in the waveform diagram.

8.2.4.2 Level settings

Access: "Menu" > "Measure" > "Advanced" tab > "Reference Level" > "Levels" tab.

On the "Levels" tab, you define how the reference levels are calculated, or you set them directly.

The following setups are available, according to the selected "Mode":

Automatic reference level mode

In automatic reference level mode, the reference levels are always relative values. You can select one of the predefined sets, or define individual percentage values.

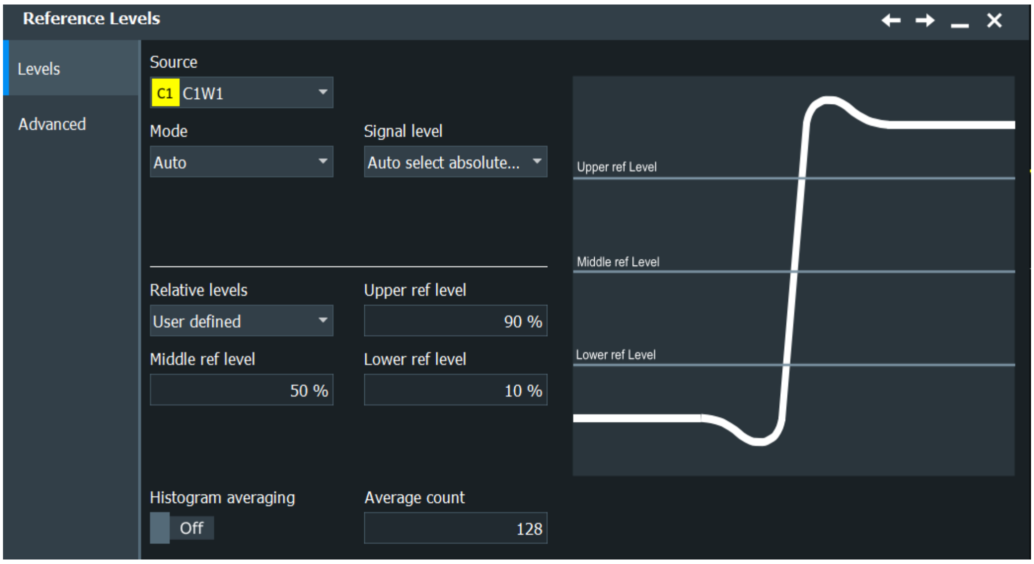


Figure 8-3: Automatic reference level definition

Manual reference level mode

In manual reference level mode, relative and absolute level definitions are possible.

In manual reference level mode with relative level definition, you define the absolute values of high and low signal levels or reference levels, and the reference levels as percentages of the signal amplitude.

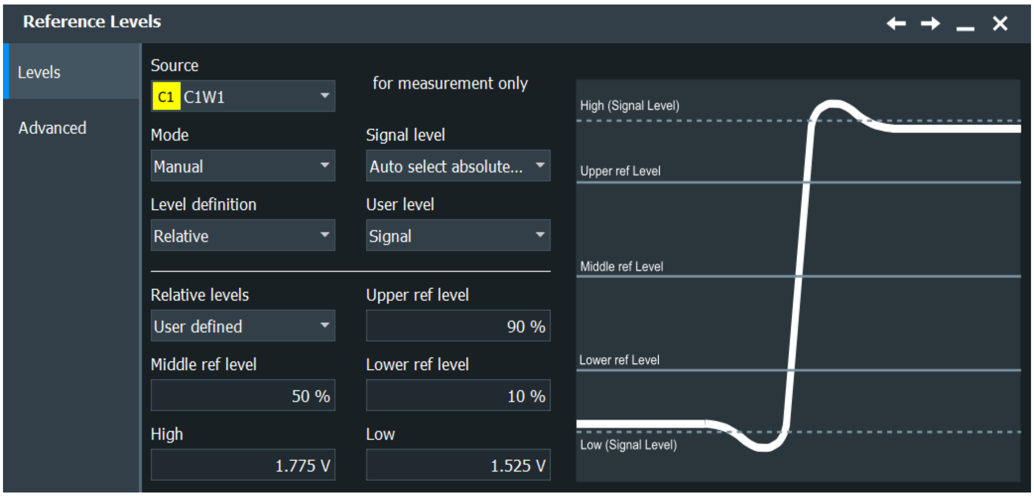


Figure 8-4: Manual reference level mode, relative level definition

In manual reference level mode with absolute level definition, you define the absolute values of high and low signal levels or reference levels, and the distances between reference and signal levels.

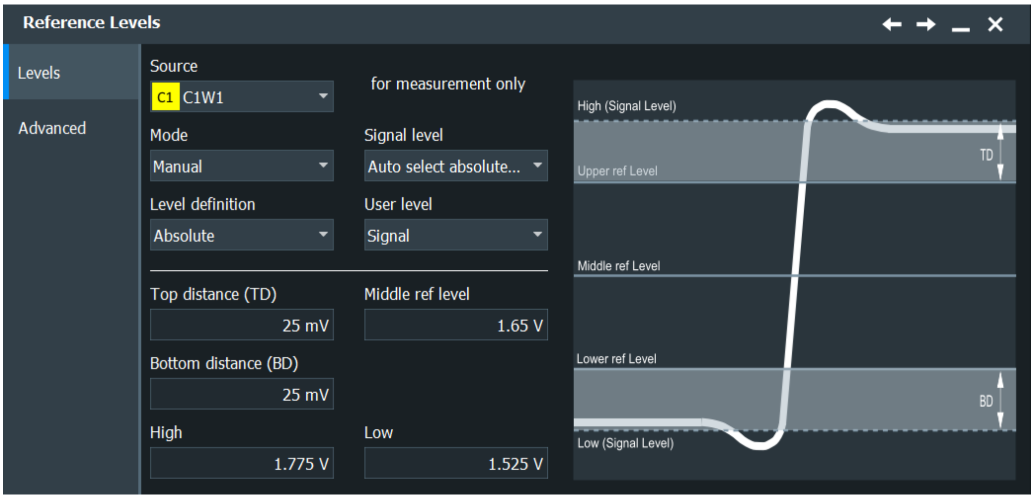


Figure 8-5: Manual reference level mode, absolute level definition

Source

Defines the source for which the reference levels are defined. The source can be any signal input, math or reference waveform.

Remote command:

Source is defined by suffix <m> in "REFLevel" subsystem, see Chapter 23.12.14, "Reference levels", on page 1550

Mode

Defines whether the reference level is configured manually or automatically.

Remote command:

`REFLevel<m>:LDETection` on page 1550

Signal level

Defines the computation method for high and low signal levels. The instrument analyzes the signal, performs amplitude and histogram measurements, and defines the signal levels using the selected method.

The selected method is used to compute the signal levels for determination of reference levels in automatic reference level mode. It is also used for high, low, and amplitude measurements in automatic and manual reference level modes.

See also: Chapter 8.2.8, "Histograms and histogram measurements", on page 343

"Auto select absolute probability"	The most suitable signal levels for the selected measurement are used.
"Peak probability"	The signal levels with the highest probability values are used. These are the upper peak value and the lower peak value of the histogram measurement.
"Mean probability"	The signal levels with mean probabilities are used.
"Absolute peak"	The absolute peak signal levels are used. These are the maximum and minimum signal values of the amplitude measurement.
"Upper absolute peak - Lower mean probability"	The high signal level is the upper absolute peak (the maximum signal level), and the low signal level is the level with the mean probability in the lower half of the histogram.
"Upper mean probability - Lower absolute peak"	The high signal level is the level with mean probability in the upper half of the histogram, and the low signal level is the lower absolute peak (the minimum signal level).
"Upper absolute peak - Lower manual"	The high signal level is the maximum result value of the amplitude measurement. The low signal level is manually set using "Low".
"Upper manual - Lower absolute peak"	The high signal level is set manually using "High". The low signal level is the minimum result value of the amplitude measurement.

Remote command:
`REFLevel<m>:AUTO:MODE` on page 1552

Level definition

In manual reference level mode, the setting defines whether the reference is configured using absolute or relative values.

Remote command:
`REFLevel<m>:LMODe` on page 1551

User level

In manual reference level mode, the setting defines whether the user-defined signal levels or user-defined reference levels are used for the measurements.

"Signal" You can define the high and low signal levels.

"Reference" You can define the reference levels.

Remote command:
`REFLevel<m>:USRLevel` on page 1552

Relative levels
Sets the lower, middle and upper reference levels, defined as percentages of the signal amplitude.

Available relative levels:

- 5/50/95
- 10/50/90
- 20/50/80
- User defined: Enter "Upper ref level", "Middle ref level", and "Lower ref level".

For example, for "5/50/95" the levels are set to the following values:

- Lower reference level = 5% of the signal amplitude
- Middle reference level = 50% of the signal amplitude
- Upper reference level = 95% of the signal amplitude

Remote command:
`REFLevel<m>:RELative:MODE` on page 1551

Upper ref level, Middle ref level, Lower ref level
Define the reference levels in percent, if "Relative levels" is set to "User-defined".

Remote command:
`REFLevel<m>:RELative:UPPer` on page 1557
`REFLevel<m>:RELative:MIDDle` on page 1558
`REFLevel<m>:RELative:LOWer` on page 1558

High
Sets the high signal level.

The high signal level is set in manual reference level mode, for absolute level definition and user signal level selection.

Remote command:
`REFLevel<m>:ABSolute:HIGH` on page 1554

Low
Sets the low signal level.

The low signal level is set in manual reference level mode, for absolute level definition and user signal level selection.

Remote command:
`REFLevel<m>:ABSolute:LOW` on page 1555

Middle
For user signal level selection, the level is the middle level between high and low signal level. The value is adjusted automatically if you change the high or low signal levels. Vice versa, if you change the middle level, the high and low signal levels are adjusted.

For user reference level selection, the level is the middle level between upper and lower reference level. The value is adjusted automatically if you change the upper or lower reference levels. Vice versa, if you change the middle level, the upper and lower reference levels are adjusted.

Remote command:
`REFLevel<m>:ABSolute:MLEVel` on page 1556

Top distance
The distance between the high signal level and the upper reference level - for manual reference level mode and absolute level definition.

Remote command:
REFLevel<m>:ABSolute:TDIStance on page 1555

Bottom distance
The distance between the lower reference level and the low signal value - for manual reference level mode and absolute level definition.

Remote command:
REFLevel<m>:ABSolute:BDIStance on page 1556

Upper level
The upper reference level required, for example, to determine a rise - for manual reference level mode, absolute level definition and user reference level.

Remote command:
REFLevel<m>:ABSolute:ULEVel on page 1556

Lower level
The lower reference level required, for example, to determine a fall - for manual reference level mode, absolute level definition and user reference level.

Remote command:
REFLevel<m>:ABSolute:LLEVel on page 1557

Histogram averaging
Enables averaging over several histograms to determine the reference levels.

This function is only available in automatic reference level mode.

Remote command:
REFLevel<m>:AUTO[:STATe] on page 1553

Average Count
Defines the number of histograms to calculate the average from.

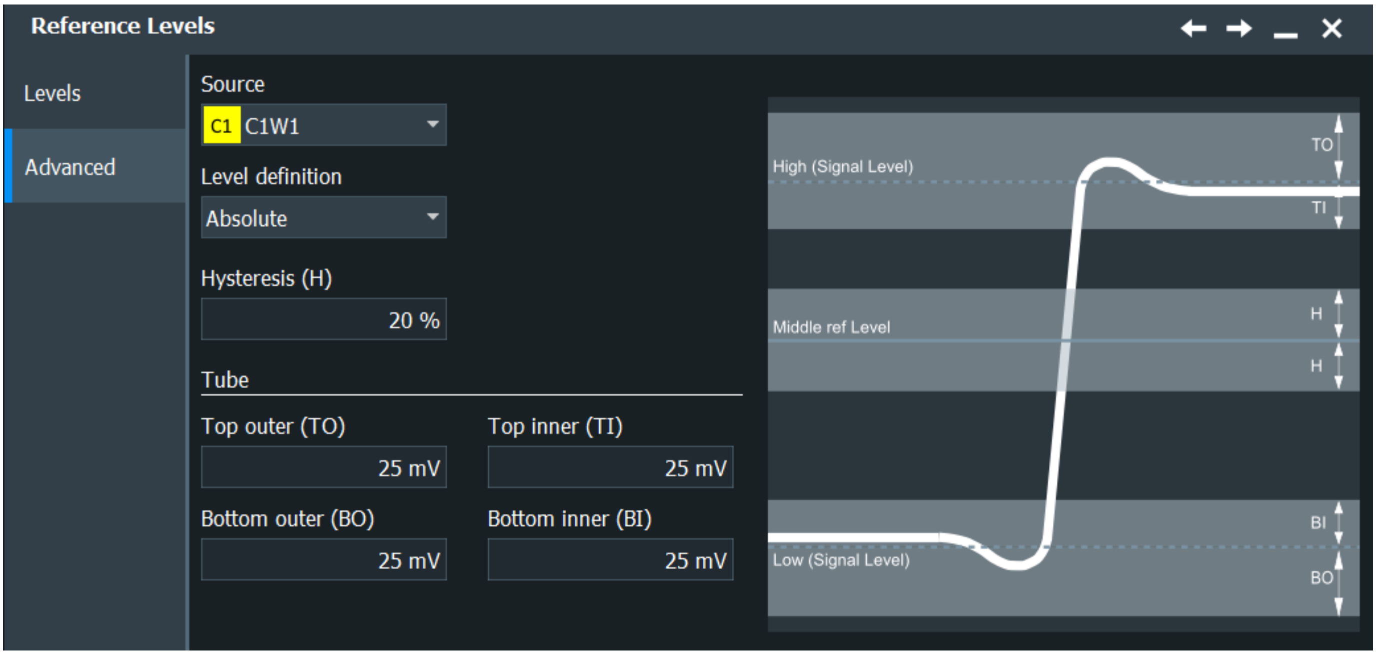
This function is only available in automatic reference level mode.

Remote command:
REFLevel<m>:AUTO:COUNt on page 1554

8.2.4.3 Advanced tab

Access: "Menu" > "Measure" > "Advanced" tab > "Reference Level" > "Advanced" tab.

This tab allows you to define evaluation tubes for measurements that require high-level or low-level detection. If the signal value remains within the defined tubes, it is considered to be high or low.



See also: Source and Level definition.

Hysteresis
Defines a hysteresis for the middle reference level. A rise or fall from the middle reference value that does not exceed the hysteresis is rejected as noise.

Remote command:
`REFLevel<m>:RELative:HYSTeresis` on page 1559

Top outer
Defines an area above the high signal level which is still considered to be high level.

Remote command:
`REFLevel<m>:ABSolute:TOTube` on page 1560
`MEASurement<m>:REFLevel:RESult:TOUTer?` on page 1563

Top inner
Defines an area beneath the high signal level which is still considered to be high level.

Remote command:
`REFLevel<m>:ABSolute:TITube` on page 1560
`MEASurement<m>:REFLevel:RESult:TINNer?` on page 1562

Bottom inner
Defines an area above the low signal level which is still considered to be low level.

Remote command:
`REFLevel<m>:ABSolute:BITube` on page 1560
`MEASurement<m>:REFLevel:RESult:BINNer?` on page 1562

Bottom outer
Defines an area beneath the low signal level which is still considered to be low level.

Remote command:
REFLevel<m>:ABSolute:BOTube on page 1561
MEASurement<m>:REFLevel:RESult:BOUTer? on page 1562

Relative outer
Defines a percentage of the signal level by which the absolute signal level may be larger than the high signal level or lower than the low signal level to be considered high or low, respectively.

Remote command:
REFLevel<m>:RELative:OTUBe on page 1559

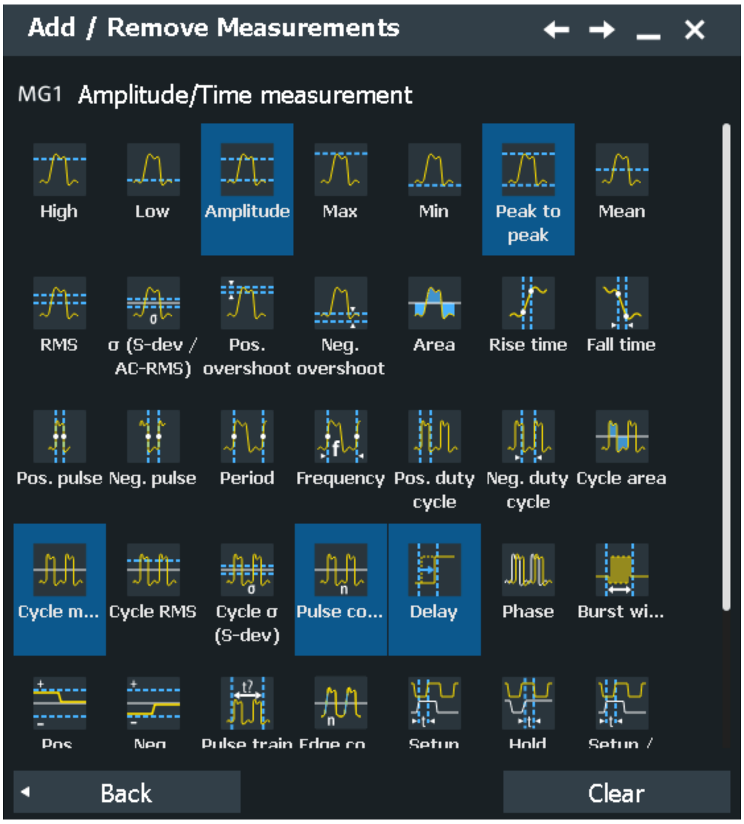
Relative inner
Defines a percentage of the signal level by which the absolute signal level may be higher than the low signal level or lower than the high signal level to be considered low or high, respectively.

Remote command:
REFLevel<m>:RELative:ITUBe on page 1559

8.2.5 Amplitude/time measurements

8.2.5.1 Overview of amplitude/time measurements

Access: [Meas] > "Meas Group" tab > "Amp/Time" category



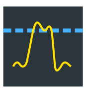
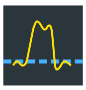
The R&S RTO provides various voltage, time, area and counting measurements in the category "Amp/Time". Some measurements require reference levels to be set according to the measurement purpose.

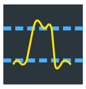
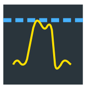
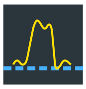
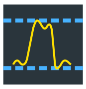
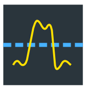
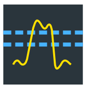
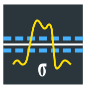
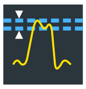
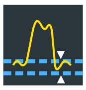
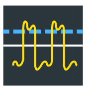
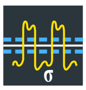
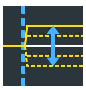
Reference levels are explained in Chapter 8.2.4, "Reference levels", on page 311.

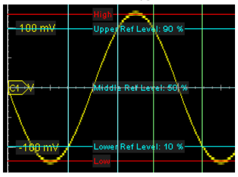
Amplitude measurements

Access: [Meas] > "Meas Group" tab > "Amp/Time" category

Table 8-1: Amplitude measurements

	Meas. type	Symbol	Description/Result
	High	X_{High}	High signal level
	Low	X_{Low}	Low signal level

	Meas. type	Symbol	Description/Result
	Amplitude	X_{Ampl}	Amplitude of the signal: the difference of high and low signal levels $X_{Ampl} = X_{High} - X_{Low}$
	Max	X_{Max}	Absolute maximum value of the waveform
	Min	X_{Min}	Absolute minimum value of the waveform
	Peak to peak	X_{PkPk}	Peak-to-peak value of the waveform: the difference of maximum and minimum values $X_{Ampl} = X_{Max} - X_{Min}$
	Mean	X_{Mean}	Arithmetic average of the waveform voltage values $X_{Mean} = \frac{1}{N_{Eval}} \sum_{i=1}^{N_{Eval}} x(i)$
	RMS	X_{RMS}	RMS (root mean square, quadratic mean) of the waveform voltage values $X_{RMS} = \sqrt{\frac{1}{N_{Eval}} \sum_{i=1}^{N_{Eval}} x^2(i)}$
	σ (S-dev)	σ_X	Standard deviation of the waveform samples $\sigma_X = \sqrt{\frac{1}{N_{Eval} - 1} \sum_{i=1}^{N_{Eval}} (x(i) - X_{Mean})^2}$
	Pos. overshoot	R_{Pos}	Positive overshoot of a square wave, calculated from measurement values High, Max, and Amplitude $+Ovr = \frac{V_{top} - V_{P+}}{V_{Amp}} \cdot 100\%$
	Neg. overshoot	R_{Neg}	Negative overshoot of a square wave, calculated from measurement values Min, Low, and Amplitude $-Ovr = \frac{V_{base} - V_{P-}}{V_{Amp}} \cdot 100\%$
	Cycle mean		The mean value of one cycle
	Cycle RMS		The RMS (root mean square) value of one cycle
	Cycle σ (S-dev)		The standard deviation of one cycle
	Trig. ProbeMeter		The DC voltage from the connected probe. Only available if an active Rohde & Schwarz probe with ProbeMeter is connected.

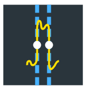
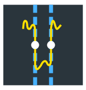
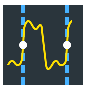
	Meas. type	Symbol	Description/Result
	Slew rate rising		Steepness of the rising edge: voltage difference between the lower and higher reference level, divided by the rise time. Result in V/s = V*Hz (blue vertical lines in the picture below).
	Slew rate falling		Steepness of the falling edge: voltage difference between the higher and lower reference level, divided by the fall time. Result in V/s = V*Hz (green vertical lines in the picture below). 

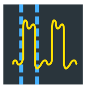
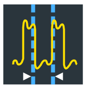
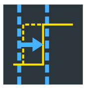
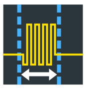
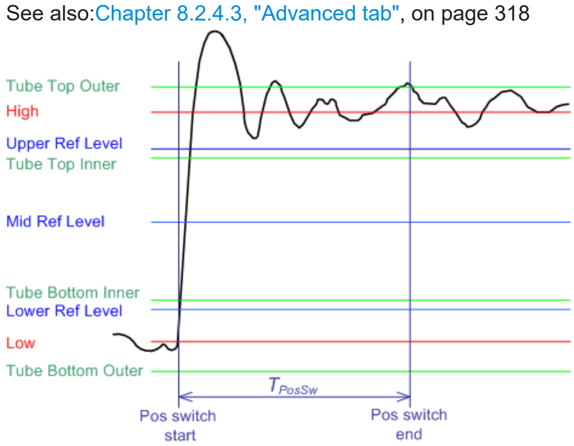
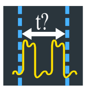
Time measurements

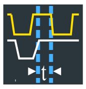
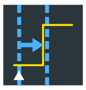
Access: [Meas] > "Meas Group" tab > "Amp/Time" category

Reference levels are explained in Chapter 8.2.4, "Reference levels", on page 311.

Table 8-2: Time measurements

	Meas. type	Symbol	Description/Result
	Rise time	T_{Rise}	Rise time of the left-most rising edge of the waveform. Rise time is the time it takes the signal to rise from the low reference level to the high reference level. Measurement all events in the acquisition is possible.
	Fall time	T_{Fall}	Falling time of the left-most falling edge of the waveform. Fall time is the time it takes the signal to fall from the high reference to the low reference. Measurement all events in the acquisition is possible.
	Pos. pulse	$T_{PosPulse}$	Width of a positive pulse: time between a rising edge and the following falling edge measured on the middle reference level. The measurement requires at least one complete period of a triggered signal. Measurement all events in the acquisition is possible.
	Neg. pulse	$T_{NegPulse}$	Width of a negative pulse: time between a falling edge and the following rising edge measured on the middle reference level. The measurement requires at least one complete period of a triggered signal. Measurement all events in the acquisition is possible.
	Period	T_{Period}	Time between two consecutive waveform edges of the same direction, measured on the middle reference level. The measurement requires at least one complete period of a triggered signal. Measurement all events in the acquisition is possible.
	Frequency	f_{Period}	Frequency of the signal, reciprocal value of the period. $f_{Period} = 1 / T_{Period}$

	Meas. type	Symbol	Description/Result
	Pos. duty cycle	R_{PosCyc}	Positive duty cycle: Width of a positive pulse in relation to the period in %. The measurement requires at least one complete period of a triggered signal. Multiple measurements are possible. $$R_{PosCyc} = \frac{T_{PosPulse}}{T_{Period}} \cdot 100\%$$
	Neg. duty cycle	R_{NegCyc}	Negative duty cycle: Width of a negative pulse in relation to the period in %. The measurement requires at least one complete period of a triggered signal. Multiple measurements are possible. $$R_{NegCyc} = \frac{T_{NegPulse}}{T_{Period}} \cdot 100\%$$
	Delay		Time difference between any two edges of two measurement sources at any reference level. The measurement result is negative if the edge of the second source comes before the edge of the first source. See: "Settings for delay and phase measurements (analog sources)" on page 327
	Phase		Phase difference between two waveforms. *Phase = Delay / Period * 360*
	Burst width		Duration of one burst, measured from the first edge to the last
	Pos. switching	T_{PosSw}	Settling time at rising edges: Time between crossing the lower reference level and the last return of the signal into the top tolerance tube. See also:Chapter 8.2.4.3, "Advanced tab", on page 318 
	Neg. switching	T_{NegSw}	Settling time at falling edges: Time between crossing the upper reference level and the last return of the signal into the bottom tolerance tube. See also "Pos. switching" above.
	Pulse train		Duration of N positive pulses, measured from the rising edge of the first pulse to the falling edge of the N-th pulse. Define N for the measurement.

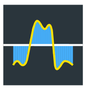
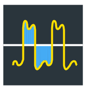
	Meas. type	Symbol	Description/Result
	Setup Hold Setup/Hold time	T_{Setup} and T_{Hold}	Setup and Hold time measurement with positive and/or negative clock edge. See: "Setup/Hold measurement settings" on page 329
	Setup/Hold ratio	T_{Setup} / (T_{Setup} + T_{Hold})	Setup/Hold ratio measurement with positive and/or negative clock edge. See: "Setup/Hold measurement settings" on page 329
	Delay to trigger		Time between the trigger event and a following signal slope. High accuracy even if the trigger event is outside the acquisition data. See: • "Delay to trigger measurement settings" on page 330

Area measurements

Access: [Meas] > "Meas Group" tab > "Amp/Time" category

Area measurements are voltage over time measurements.

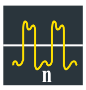
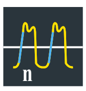
Table 8-3: Area measurements

	Meas. type	Symbol	Description/Result
	Area	A_{Ref}	Area between the waveform and a reference level ("Area level", X_{Ref}). $A_{Ref} = \frac{T_{Eval}}{N_{Eval}} \cdot \sum_{i=1}^{N_{Eval}} (x(i) - X_{Ref})$ T_{Eval}: Evaluation time, time of a full waveform or limited by a gate
	Cycle area	A_{RefCyc}	Area between the waveform and a reference level ("Area level") measured for one period, see also "Area" measurement. The measurement requires at least one complete period of a triggered signal. Multiple measurements are possible.

Counting

Access: [Meas] > "Meas Group" tab > "Amp/Time" category

Table 8-4: Counting measurements

	Meas. type	Symbol	Description/Result
	Pulse count		The number of positive or negative pulses of the waveform, or of both positive and negative pulses. The mean value of the signal is determined. If the signal passes the mean value, an edge is counted. A positive pulse is counted if a rising edge and a following falling edge are detected. A negative pulse is counted if a falling edge and a following rising edge are detected.
	Edge count		The number of positive or negative edges, or of both positive and negative edges. The instrument determines the mean value of the signal and counts an edge every time the signal passes the mean value.

Arithmetic

Access: [Meas] > "Meas Group" tab > "Amp/Time" category

Table 8-5: Arithmetic measurements

	Meas. type	Symbol	Description/Result
	Arithmetic		Allows you to perform basic arithmetic operations between selected measurement values and/or constant.

8.2.5.2 Settings for amplitude/time measurements

Access: [Meas] > "Meas Group" tab > "Amp/Time" category

Amplitude and time measurements are available for sources in the time domain. For some amplitude/time measurements, such as delay, setup/hold and delay to trigger, further setting are required to get a measurement result.

If further settings are available for a measurement, a settings icon is shown beside the measurement's name in the "Active measurements" list. Tap the icon to configure the measurement.

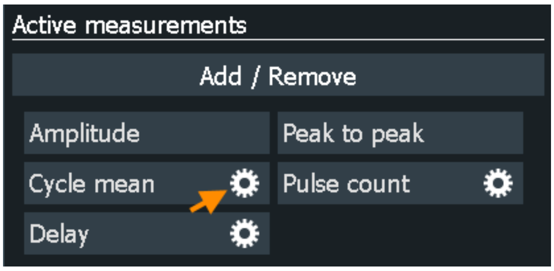


Figure 8-6: Selected active measurements, some have further settings

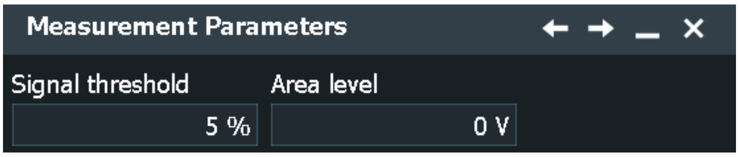


Figure 8-7: Settings for cycle area measurement

This chapter explains all settings for amplitude/time measurements. For a description of available measurement, see Chapter 8.2.5.1, "Overview of amplitude/time measurements", on page 320.

Signal threshold

Defines a signal value that must be exceeded for the signal value to be included in the measurement. The setting is relevant for area, time, and counting measurements.

Remote command:
`MEASurement<m>:DETThreshold` on page 1494

Area level

The reference level used to integrate the waveform. The setting is only relevant for area measurements.

Remote command:
`MEASurement<m>:AMPTime:ALEVel` on page 1494

Pulses slope

Sets the first slope of the pulses to be counted.

The setting is available only for the "Pulse count" measurement.

"Positive"	Positive pulses are counted.
"Negative"	Negative pulses are counted.
"Either"	Both positive and negative pulses are counted.

Remote command:
`MEASurement<m>:AMPTime:PSLope` on page 1495

Measured slope

Selects the slope direction for frequency and period measurements.

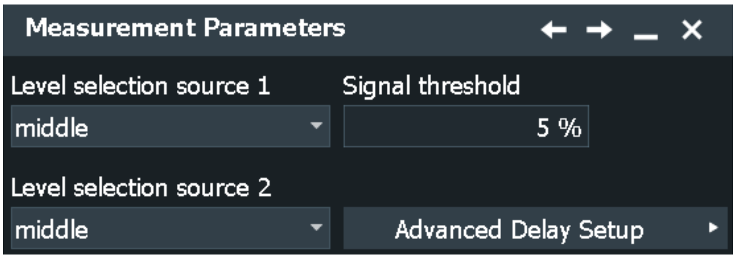
"Positive / Negative"	Measures the time between rising or falling edges, respectively.
"Either"	In multiple measurements, the time is measured both between rising edges and between falling edges. In single measurements. The first edge is taken for the measurement.
"First edge"	Time is measured either between rising edges or between falling edges. The first edge is taken for the measurement. In single measurements, it works the same way as "Either". Only available for analog measurement sources.

Remote command:
`MEASurement<m>:AMPTime:PFSLope` on page 1495

Settings for delay and phase measurements (analog sources)

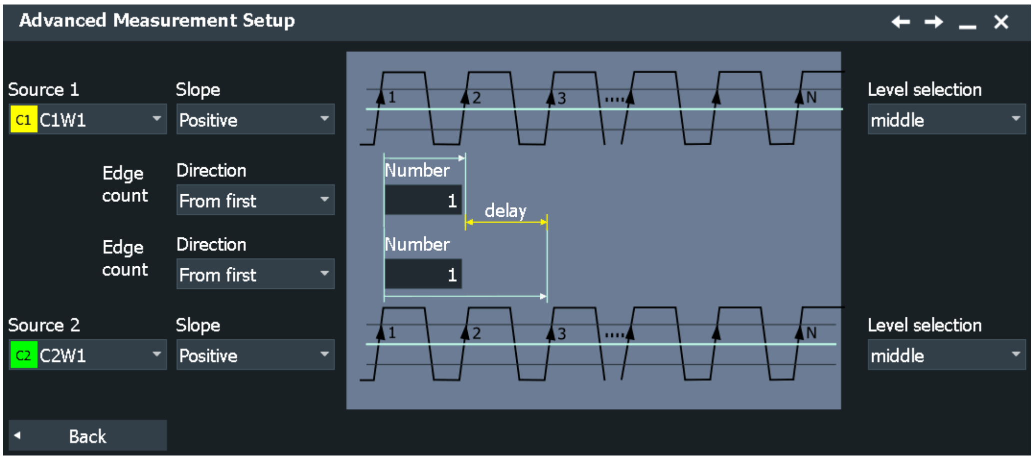
The specific settings for delay measurement allow you to measure the time between any two slopes at any reference level. Therefore, the reference levels and the slopes must be defined for each source individually. The measurement result is negative if the edge of the second source comes before the edge of the first source.

"Level selection source 1/2": For each measurement source, select the reference level on which the time is measured.



"Advanced Delay Setup":

With the settings shown in the picture, the time between the second and the fifth rising edge is measured.



"Level selection"	Selects the reference level on which the time is measured.
"Slope"	Sets the edge of each source, between which the delay is measured: positive, negative, or either of them.
"Direction"	Selects the direction for counting slopes for each source: from the beginning of the waveform, or from the end.
"Number"	Sets the number of the edge that is relevant for delay measurement.

Remote command:
`MEASurement<m>:AMPTime:DELay<n>:LSELect` on page 1496
`MEASurement<m>:AMPTime:DELay<n>:SLOPe` on page 1497
`MEASurement<m>:AMPTime:DELay<n>:DIRection` on page 1496
`MEASurement<m>:AMPTime:DELay<n>:ECOunt` on page 1496

Settings for delay and phase measurements (digital sources)

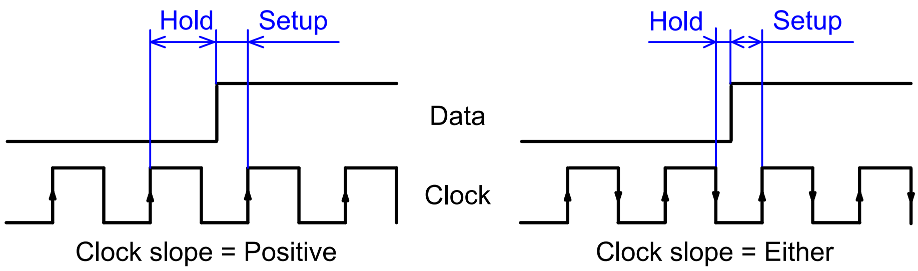
Delay measurement on digital channels is reduced to measure the time between two subsequent rising or two subsequent falling edges.

The edge direction is set with Edges slope.

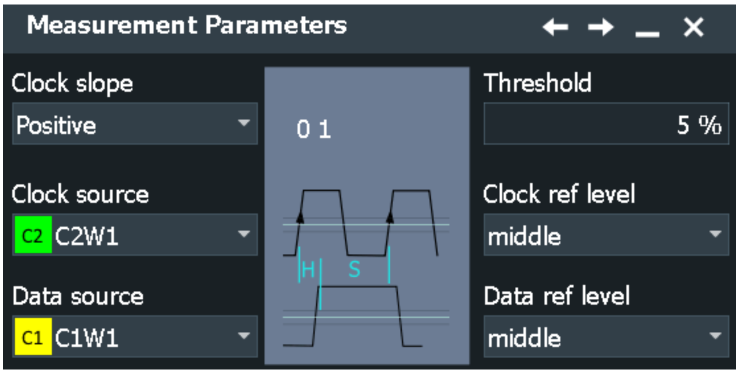
Setup/Hold measurement settings
Setup/Hold measurements analyze the relative timing between two signals: a data signal and the synchronous clock signal. Setup time is the time that the data signal is steady before clock edge - the time between a data transition and the next specified clock edge. Hold time is the time that the data signal is steady after clock edge - the time between a data transition and the previous specified clock edge.

"Setup/Hold time" measures and displays the setup and hold durations. "Setup/Hold ratio" measurements return the ratio of the setup time to the sum of hold and setup time: $T_{Setup} / (T_{Setup} + T_{Hold})$.

The clock edge can be defined, the polarity of the data signal does not matter.



If at least one of the setup/hold measurements is selected, more settings appear to specify the measurement.



"Clock slope" Sets the edge of the clock from which the setup and hold times are measured: positive, negative, or either of them. If "Either" is selected, the clock edges next to the data edge are considered regardless of the clock slope.

"Clock source" The "Clock source" is identical to the measurement "Source". It defines the waveform used as clock in the setup/hold measurement.

"Data source" The "Data source" is identical to the "2nd Source" of the measurement. It sets the data signal.

"Clock ref level" See "Clock ref level" on page 330.

"Data ref level" See "Data ref level / Reference level" on page 330.

"Threshold" See "Signal threshold" on page 327.

Remote command:
Clock slope: `MEASurement<m>:AMPTime:CSLope` on page 1498

Clock ref level

Selects the reference level of the clock on which the time is measured. The intersection of slope and reference level defines the time point for measurements.

The setting is used for setup and hold measurements, and for jitter measurements (option R&S RTO-K12).

Remote command:
`MEASurement<m>:AMPTime:CLCK<n>:LSELect` on page 1498

Data ref level / Reference level

The setting defines the "Data ref level" for setup and hold measurements, and for time-interval error measurements (option R&S RTO-K12). It selects the reference level of the data signal on which the time is measured. The intersection of slope and reference level defines the time point for measurements.

For clock jitter measurements (option R&S RTO-K12), it sets the "Reference level" for the time measurement.

Remote command:
`MEASurement<m>:AMPTime:DATA<n>:LSELect` on page 1499

Pulse count

Sets the number N of positive pulses for the "Pulse train" measurement. This measurement measures the duration of N positive pulses from the rising edge of the first pulse to the falling edge of the N-th pulse.

Remote command:
`MEASurement<m>:AMPTime:PTCount` on page 1497

Edges slope

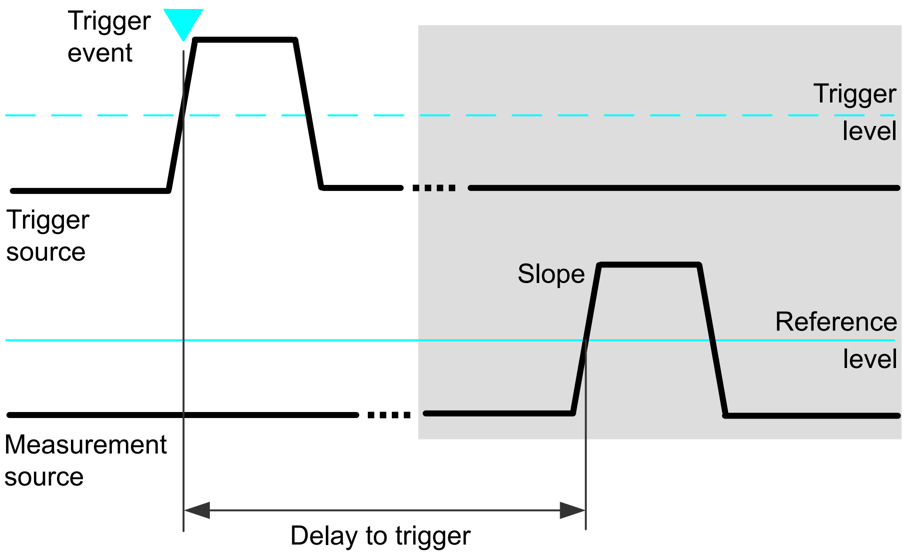
Sets the edge direction to be considered. The setting is relevant for edge count measurement and delay measurement on digital channels.

"Positive"	Positive edges are considered.
"Negative"	Negative edges are considered.
"Either"	Both positive and negative edges are counted (edge count). Delay is measured either between rising edges or between falling edges. The first edge is taken for the measurement.

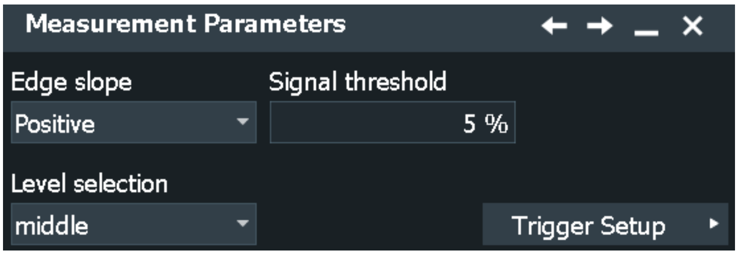
Remote command:
`MEASurement<m>:AMPTime:ESLope` on page 1498

Delay to trigger measurement settings

Delay to trigger measures the time between the trigger point and the following slope of a waveform. The delay between the trigger and the slope can be high compared to the accuracy of the acquisition, and the trigger point can even be outside of the current acquisition.



To configure the trigger conditions, use the trigger setup. To set up the slope, additional settings appear in the measurements "Setup" dialog box.



"Edge slope" Sets the edge direction to be used for delay measurement: positive, negative, or either edge.

"Level selection" Selects the reference level of the measurement source on which the delay is measured: upper, middle, or lower level.

"Signal threshold"
See "Signal threshold" on page 327.

Remote command:
`MEASurement<m>:AMPTime:DTOTrigger<n>:SLOPe` on page 1499
`MEASurement<m>:AMPTime:DTOTrigger<n>:LSELect` on page 1499

8.2.5.3 Arithmetic for amplitude/time and spectrum measurements

Access: [Meas] > "Meas Group" tab > "Amp/Time"/"Spectrum" category > "Arithmetic".

In this dialog, you can set up basic arithmetic operations between selected measurement values and/or constant.

You can select only measurement groups and measurements that are already enabled.

Type
Selects the type of arithmetic measurement:

"Measure-ment" Performs an arithmetic operation between two measurement results. You can select from the measurements enabled for "Amp/Time" or "Spectrum".

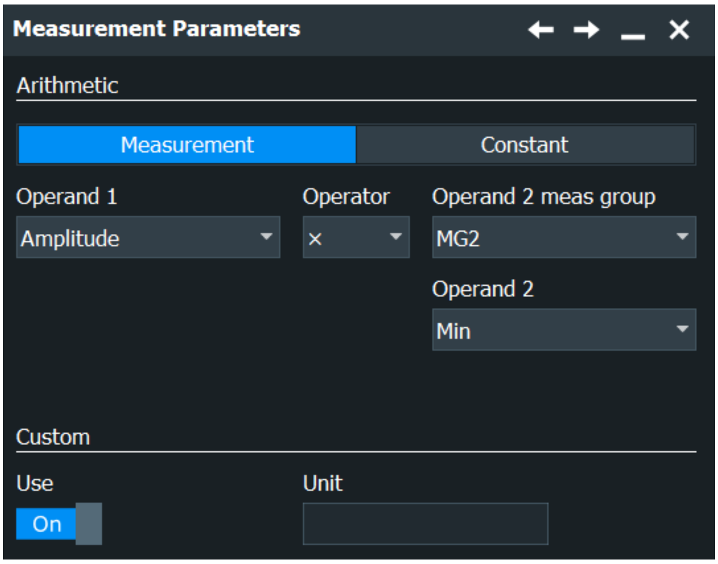


Figure 8-8: Arithmetic with two measurement results

"Constant" Performs an operation between a measurement result and a user defined fixed Constant.

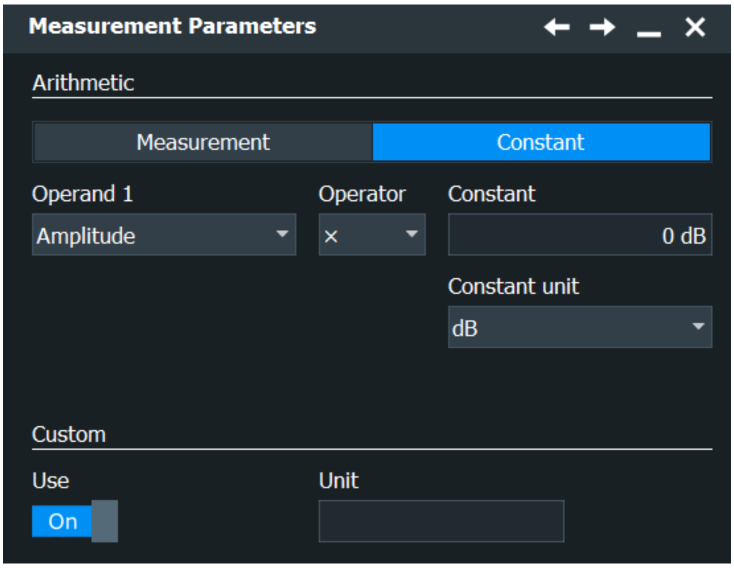


Figure 8-9: Arithmetic with a measurement result and a constant

Remote command:
`MEASurement<m>:AMPTime:ARIThmetic:PARameter<n>:TYPE` on page 1505

Operand 1
Selects the first operand for the arithmetics. You can select one of the measurements, that is enabled for the current measurement group.

Remote command:
MEASurement<m>:SPECtrum:ARIThmetic:PARameter<n>:AMPTime on page 1508
MEASurement<m>:AMPTime:ARIThmetic:PARameter<n>:AMPTime on page 1504
MEASurement<m>:AMPTime:ARIThmetic:PARameter<n>:SPECtrum on page 1505
MEASurement<m>:SPECtrum:ARIThmetic:PARameter<n>:SPECtrum on page 1509

Operand 2 meas group / Operand 2
Selects the measurement group and the measurement result for the second operand. You can select one of the measurements, that is enabled for the selected measurement group.

Remote command:
MEASurement<m>:AMPTime:ARIThmetic:PARameter<n>:GROup on page 1504
MEASurement<m>:SPECtrum:ARIThmetic:PARameter<n>:GROup on page 1508

Operator
Selects the operation type to be performed on the selected operands. You can choose between +, -, * and /.

"+"	Adds up the operands.
"-"	Subtracts "Operand 2"/"Constant" from "Operand 1".
"*"	Multiplies the operands.
"/"	Divides "Operand 1" by "Operand 2"/"Constant".

Remote command:
MEASurement<m>:AMPTime:ARIThmetic:OPERator on page 1503

Custom
Enable "Use" to apply the user-defined value for the "Unit".

Remote command:
MEASurement<m>:AMPTime:ARIThmetic:CUNState on page 1502
MEASurement<m>:SPECtrum:ARIThmetic:CUNState on page 1506

Constant
Sets a constant numeric value for the arithmetic operation, for the "Constant" measurement type.

Remote command:
MEASurement<m>:AMPTime:ARIThmetic:CONStant on page 1501
MEASurement<m>:SPECtrum:ARIThmetic:CONStant on page 1505

Constant unit
Selects the unit for the Arithmetic for amplitude/time and spectrum measurements.

Remote command:
`MEASurement<m>:AMPTime:ARIThmetic:COUNit` on page 1502
`MEASurement<m>:SPECtrum:ARIThmetic:COUNit` on page 1506

8.2.5.4 Measuring the delay to trigger

Delay to trigger measures the time between the trigger point and the following slope of a waveform. If the delay is unknown, it can be measured in two stages - first a coarse and then a precise measurement.

See also: "Delay to trigger measurement settings" on page 330.

1. Set the horizontal scale and horizontal position so that the trigger point and the slope both are visible on the screen.
2. Select the delay to trigger measurement:
   a) In the "Meas Group" dialog, set the "Category" to "Amp/Time".
   b) Press "Add/ Remove Measurements" button.
   c) In the dialog, enable "Delay to trigger".
   d) Tap "Back".
3. Configure the "Delay to trigger" measurement:
   a) In the "Active Measurements" list, tap the "Delay to trigger" measurement.
   b) Select the source, that is the waveform with the delayed slope.
   c) Select the slope and the reference level.
   d) Check the trigger settings.
4. Enable the measurement. Note the result.
5. Turn the horizontal [Position] knob and enter the measured delay as horizontal position.

   Thus, the slope is moved to the center of the screen.
6. Adjust the time scale ("Menu" > "Horizontal" > "Setup" tab) and the sample rate and resolution ("Menu" > "Acquire" > "Setup" tab) to the required resolution.

   The trigger is outside the display and is not part of the current acquisition.
7. Repeat the "Delay to trigger" measurement.

   Now the delay is measured with high accuracy. You can analyze the variance of delay values using statistical evaluation and histogram functions.

8.2.6 Eye diagram measurements

The eye diagram is a tool for evaluation of signal quality and shows the combined effects of channel noise and intersymbol interference. It is a significant means of visualizing jitter and allows you to analyze the reasons for it. By creating histograms of the eye diagram, important jitter parameters can be determined.

The eye diagrams are a superposition of repetitively sampled waveforms, which have a length of about 1 bit.

The waveform display style must be set to vectors: "Menu" > "Settings" > "Display" > "Signal" > "Waveform style" = "Vectors".

To obtain optimized settings for an eye measurement, use the "Autoset" function that is provided on the right side of the "Eye" tab.

The following characteristic values can be determined:

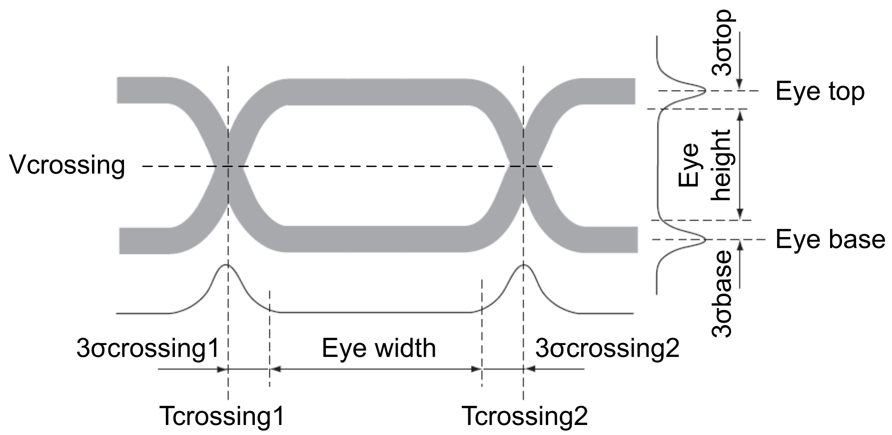


Figure 8-10: Basic eye diagram characteristics

Eye top	= Mean of the upper vertical histogram
σtop	= Standard deviation of the upper vertical histogram
Eye base	= Mean of the lower vertical histogram
σbase	= Standard deviation of the lower vertical histogram
Tcrossing	= First and second mean of the horizontal histogram
σcrossing	= Standard deviation of the horizontal histogram

8.2.6.1 Overview of eye diagram measurements

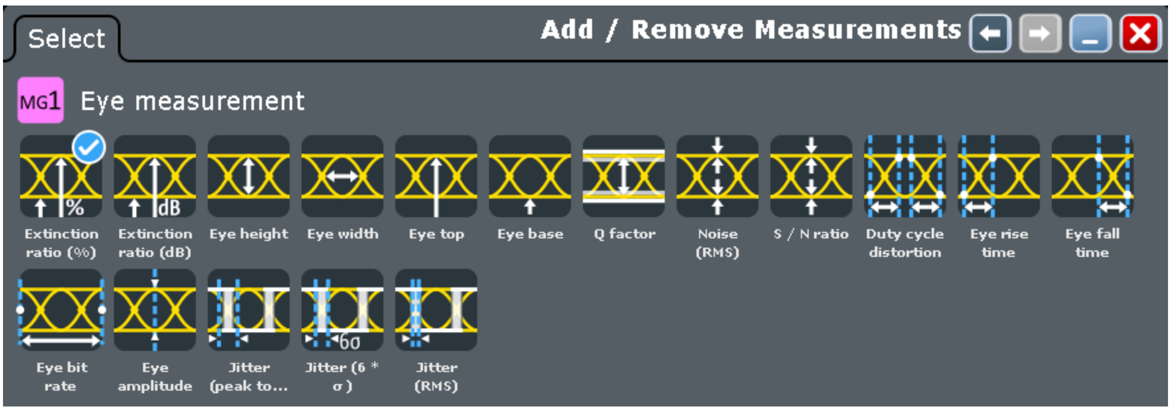
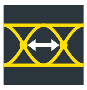
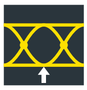
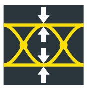
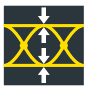
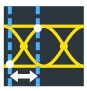
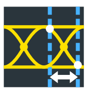
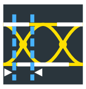
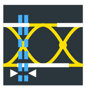


Table 8-6: Eye measurements

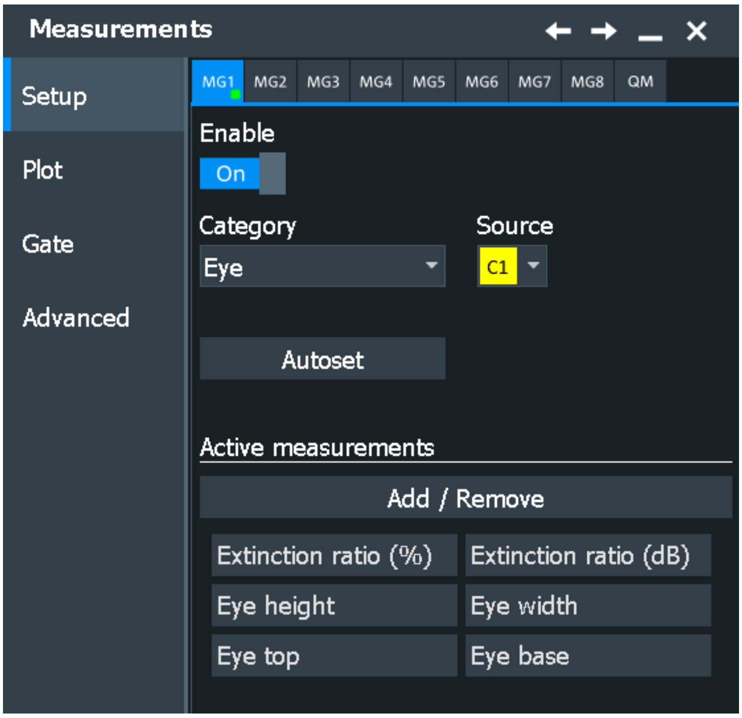
	Meas. type	Description/Result
	Extinction ratio (%)	The extinction ratio is an indication of efficiency. It describes the ratio of the power used to transmit a logic level 1, to the energy used to transmit a logic level 0. The R&S RTO provides extinction ratio measurements as a percentage, and in decibels: ER (%) = Eye base / Eye top *100 Prerequisite: Eye base > 0 and Eye top > 0 because extinction ratio is defined only for positive values.
	Extinction ratio (dB)	ER (dB) = 10*log (Eye top / Eye base)
	Eye height	The vertical eye opening indicates the sensitivity of the transmission to noise. (Eye top – 3 * σtop) – (Eye base + 3 * σbase)
	Eye width	The horizontal eye opening indicates the time range during which the sampling of the logical state is possible. (Tcrossing2 – 3 * σcrossing2) – (Tcrossing 1 – 3 * σcrossing1)
	Eye top	Mean of the upper vertical histogram
	Eye base	Mean of the lower vertical histogram
	Q factor	Q factor = (Eye top – Eye base) / (σtop + σbase)
	Noise (RMS)	Quadratic mean of the noise of eye top and eye base
	S/N ratio	Signal-to-noise ratio SNR = 10 * log (Eye amplitude / Noise RMS)
	Duty cycle distortion	Duty cycle distortion = 20 * log (Eye amplitude / Noise RMS)
	Eye rise time	Duration for signal to rise from 10% to 90% of the high signal level
	Eye fall time	Duration for signal to fall from 90% to 10% of the high signal level
	Eye bit rate	Frequency between two crossings

	Meas. type	Description/Result
	Eye amplitude	Eye top - Eye base
	Jitter (peak to peak)	Average of the jitter for both crossing points. Jitter = (σcrossing1 + σcrossing2) / 2
	Jitter (6*σ)	Jitter (6*σ) = Jitter * 6
	Jitter (RMS)	Quadratic mean of the jitter at both crossing points

8.2.6.2 Settings for eye diagram measurements

Access: [Meas] > "Meas Group" tab > "Eye" category

Eye diagram measurements are only available for sources in the time domain.



To obtain optimized settings for an eye measurement, use the "Autoset" function that is provided on the right side of the tab.

Autoset
Defines optimized settings for eye diagram measurements on the selected source.

Remote command:
`MEASurement<m>:EYEJitter:AUToscale` on page 1511

8.2.7 Spectrum measurements

Spectrum analysis determines the frequencies of a given input signal over time. Various measurements can then be performed based on the signal spectrum.

8.2.7.1 Overview of spectrum measurements

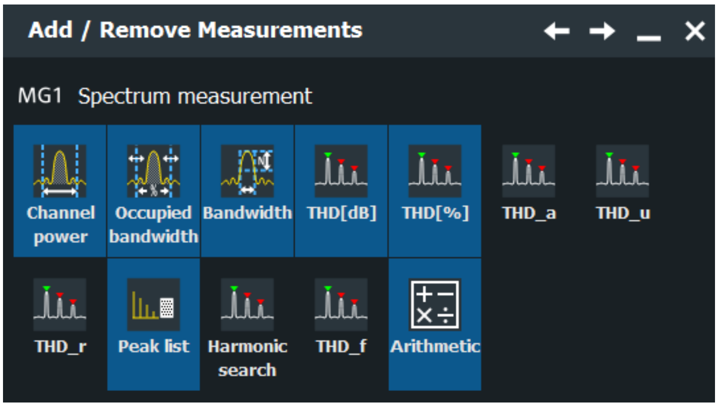
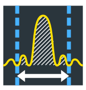
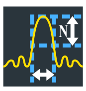
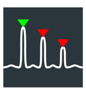
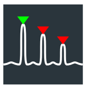
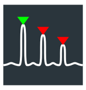
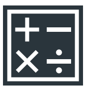


Table 8-7: Spectrum measurements

	Meas. type	Description, result
	Channel power	Power integrated over the sample values defined by a center frequency and a bandwidth; based on a defined impedance. The result is given in dBm. To get best results, enable "Measure all events in each acquisition" on page 363, and set the "Limit" to the maximum value. In particular, these settings are important when measuring pulsed signals.
	Occupied bandwidth	From the defined center frequency, symmetric sample value pairs to the left and right are integrated until a user-defined percentage of the total power is reached. The occupied bandwidth is the difference between the frequencies at which the requested power was reached.
	Bandwidth	n dB down bandwidth; the samples to the left and right of the peak value are analyzed until the n dB threshold is exceeded. The frequencies at which the threshold is exceeded define the limits of the requested bandwidth.
	THD[dB], THD[%] Total harmonic distortion	Power sum of the harmonic waves divided by the power of the fundamental wave: $THD = \frac{\sum_{n=2}^{\infty} P_n}{P_1}$

	Meas. type	Description, result
	THD_f, THD_a, THD_u, THD_r Total harmonic distortion	These measurements require option R&S RTO-K18 Spectrum Analysis. THD_f is the root mean square of the sum of all amplitudes of the harmonic waves in relation to the amplitude at the fundamental frequency (first harmonic): $THD_F = \frac{\sqrt{\sum_{i=2}^{n} U_i^2}}{U_1}$ THD_a corresponds to THD[dB]: $THD_a = \frac{\sum_{i=2}^{n} U_i^2}{U_1^2}$ THD_u: $THD_u = \frac{\sqrt{U^2 - U_1^2}}{U_1}$ Distortion factor: $THD_R = \frac{\sqrt{U^2 - U_1^2}}{U}$ Where: • U_i: effective value of the harmonic with index i • U_1: effective value of the first harmonic • U: effective value of the signal
	Peak list	This measurement requires option R&S RTO-K18 Spectrum Analysis. Table with measured peaks. For each peak, the frequency and the power value are listed in a table row. The number of determined peaks can be defined. You can sort the results by frequency or power value, and the peak labels are adjusted accordingly.
	Harmonic search	Table with measured harmonics. For each harmonic, the frequency and the value are listed in a table row.
	Arithmetic	Allows you to perform basic aritmetic operations between selected measurement values and/or constant. For details see: Chapter 8.2.5.3, "Arithmetic for amplitude/time and spectrum measurements", on page 331

For remote command parameters and suffix types, see Table 23-8.

8.2.7.2 Settings for spectrum measurements

Access: [Meas] > "Meas Group" tab > "Spectrum" category

Spectrum measurements require a source in the frequency domain, i.e. a math waveform with an FFT operation.

For spectrum measurements, make sure that the start frequency and other FFT parameters are set correctly, and the fundamentals are not covered by the DC component of the signal. Consider also a gated measurement if the instrument cannot return any result.

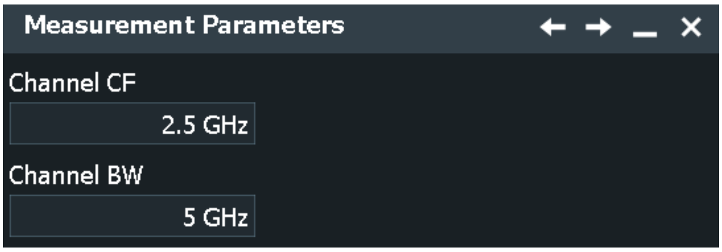


Figure 8-11: Parameters of channel power measurement

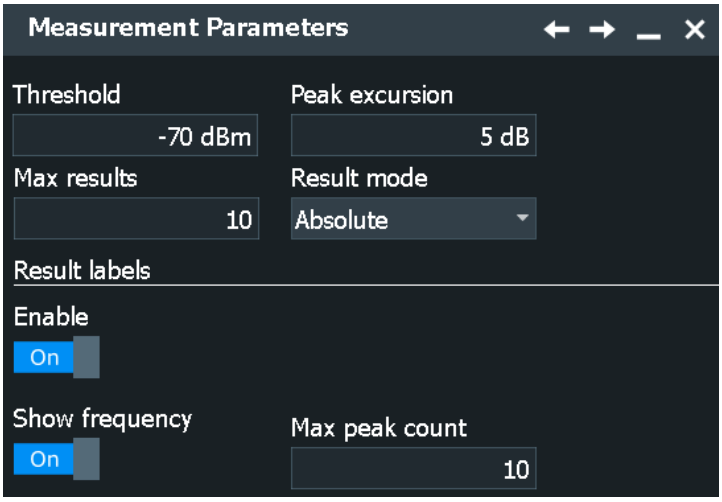


Figure 8-12: Parameters of peak list measurement (with option R&S RTO-K18)

Enable
Enables the peak list measurement for the spectogram.

N db down
The threshold until which the samples to the left and right of the peak value are analyzed to determine the "Bandwidth".

Remote command:
`MEASurement<m>:SPECtrum:NDBDown` on page 1515

Channel BW
Bandwidth over which the channel power is calculated.

Remote command:
`MEASurement<m>:SPECtrum:CPOWer:BANDwidth` on page 1514

Channel CF
Center frequency from which the channel power is calculated over the specified bandwidth.

Remote command:
`MEASurement<m>:SPECtrum:CPOWer:CFRequency` on page 1514

Occup. BW
Percentage of the total power used to determine the occupied bandwidth.

Remote command:
`MEASurement<m>:SPECtrum:OBANdwidth` on page 1514

Threshold
Defines an absolute threshold as an additional condition for the peak search. Only peaks that exceed the threshold are detected.

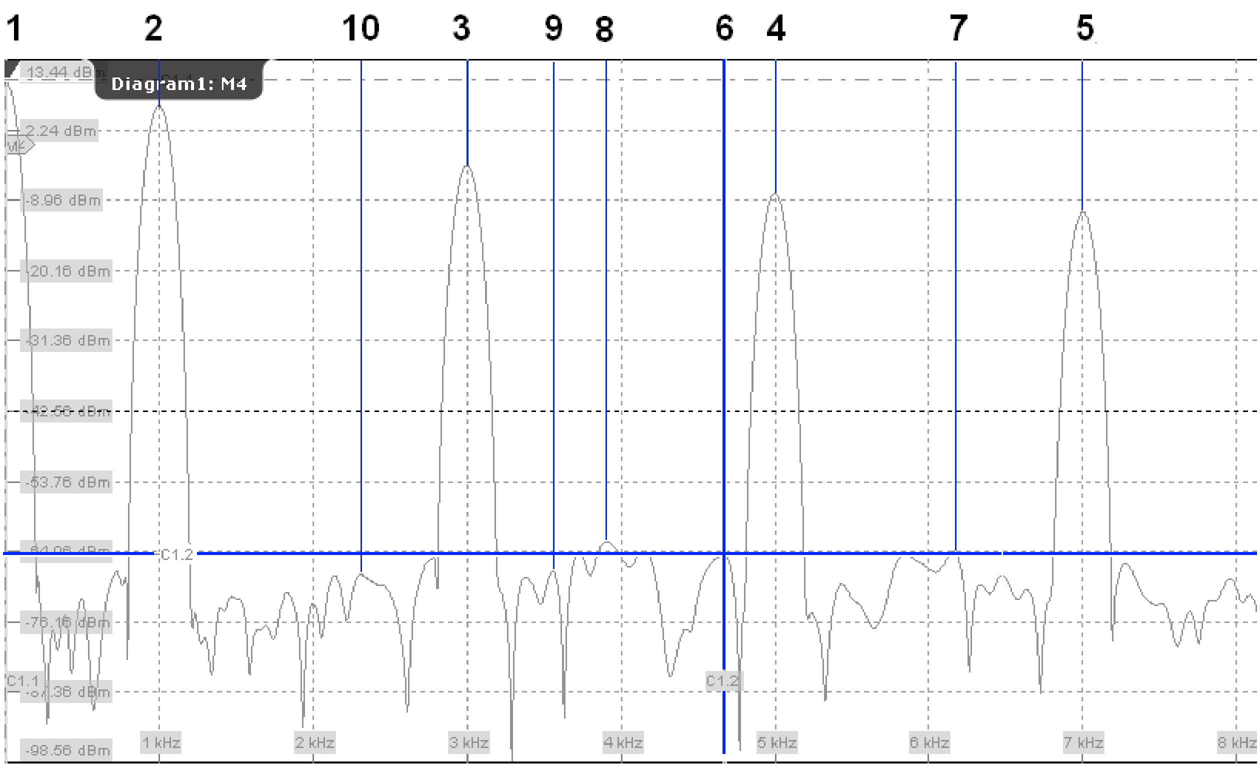
This setting is only available for spectrum waveforms. It is valid for cursor measurements, spectrum measurements and peak search.

Remote command:
`CURSor<m>:THReshold` on page 1629
`MEASurement<m>:SPECtrum:ATHReshold` on page 1515

Peak excursion
Defines a relative threshold, the minimum level value by which the waveform must rise or fall to be considered as a peak. To avoid identifying noise peaks, enter a peak excursion value that is higher than the noise levels.

This setting is only available for spectrum waveforms. It is valid for cursor measurements, spectrum measurements and peak search.

The following figure shows a cursor measurement on a spectrum waveform:



If "Peak excursion" is 30 dB, the peaks 1 to 5 are found. If "Peak excursion" is 20 dB, also the peaks 6 to 10 are found. The cursor position is on peak 6.

Remote command:
`CURSor<m>:PEXCursion` on page 1628
`MEASurement<m>:SPECtrum:PEXCursion` on page 1515

Max results
Sets the maximum number of measurement results that are listed in the result table. Available for peak list and harmonic search measurements.

Remote command:
`MEASurement<m>:SPECtrum:RESult<n>:COUNt` on page 1516

Result mode
Selects the the way the measurement results are displayed. Available for peak list and harmonic search measurements.

"Absolute" The harmonics/peaks are shown in absolute value, dBm.

"Relative" The level of the carrier is shown in absolute value dBm. The values the harmonics/peaks are shown relatively to the carrier in dBc.

Remote command:
`MEASurement<m>:SPECtrum:RESult<n>:MODE` on page 1516

Result labels
For peak lists only, requires option R&S RTO-K18: Use labels to describe the detected peaks in the spectrum diagram.

You can configure what is displayed in the labels. To change the label design, use "Menu" > "Settings" > "Appearance" > "Peak List", see Chapter 4.3.9, "Peak list appearance settings", on page 92.

"Enable" Displays a description for each detected peak in the spectrum diagram.

"Max. peak count" Defines the maximum number of peaks that are labeled in the diagram. The result table lists all peaks.

"Show Frequency" Includes the frequency of the detected peak in the diagram labels.

Remote command:
`MEASurement<m>:RESult:SHLabels` on page 1517
`MEASurement<m>:RESult:LABorder` on page 1315
`MEASurement<m>:RESult:INVerse` on page 1315
`MEASurement<m>:RESult:MAXCount` on page 1516
`MEASurement<m>:RESult:SHFRequency` on page 1517

8.2.8 Histograms and histogram measurements

8.2.8.1 Histogram characteristics

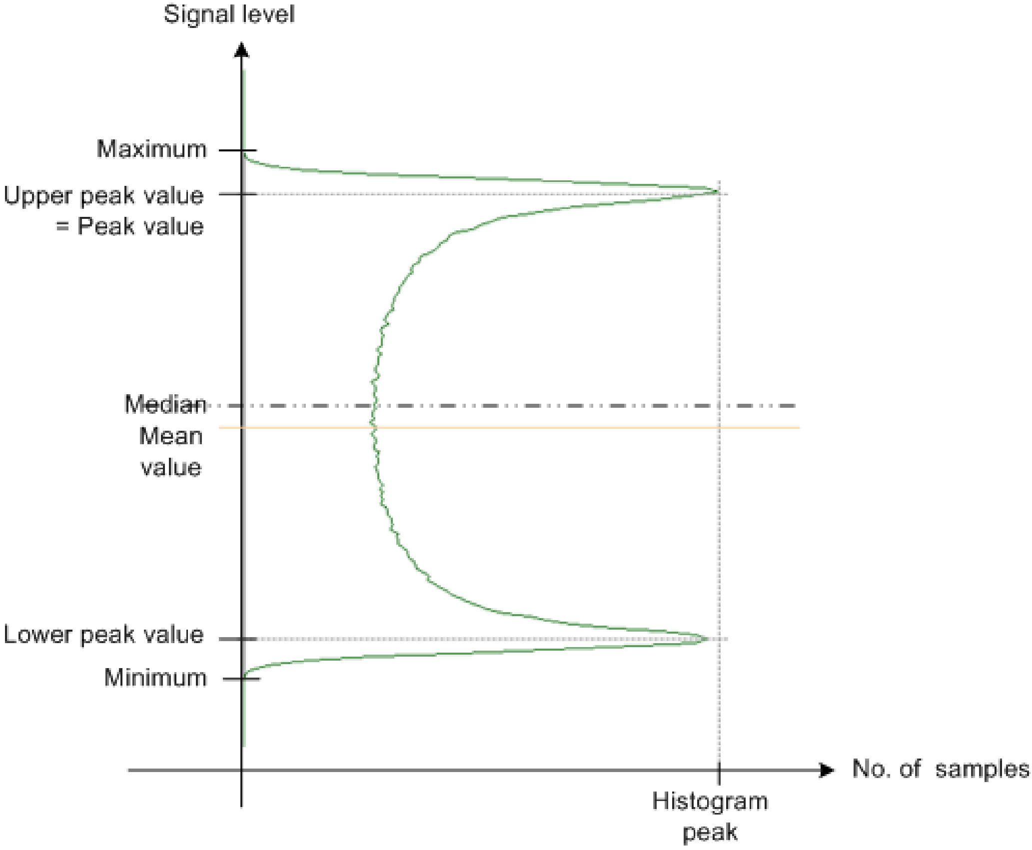
Histograms are used to plot density of data, i.e. to display graphically how often which signal values occur. The histogram can be based on the input signal levels (amplitudes) or the timebase in a time domain measurement, or on frequencies or frequency levels in a spectrum measurement. They are a prerequisite for histogram measurements.

Depending on which data the histogram is based on, a vertical or horizontal histogram can be selected. A vertical, or amplitude, histogram displays horizontal bars across amplitude values. A horizontal or time/frequency histogram displays vertical bars over time/frequencies.

You can define up to 8 histograms in a diagram, one of them is displayed. They can be created quickly using toolbar icons, or in the "Meas" menu > "Histogram" dialog box. To switch the histogram display, tap the required histogram area, or select it in the "Histogram" dialog box. For histogram measurements, the measured histogram is selected independently in the measurement setup.

In a histogram, the maximum count of a waveform value is assigned to the full height (histogram peak). All other count values are displayed relative to the maximum.

The following characteristic values can be determined for histograms (illustrated for a vertical histogram):



8.2.8.2 Overview of histogram measurements

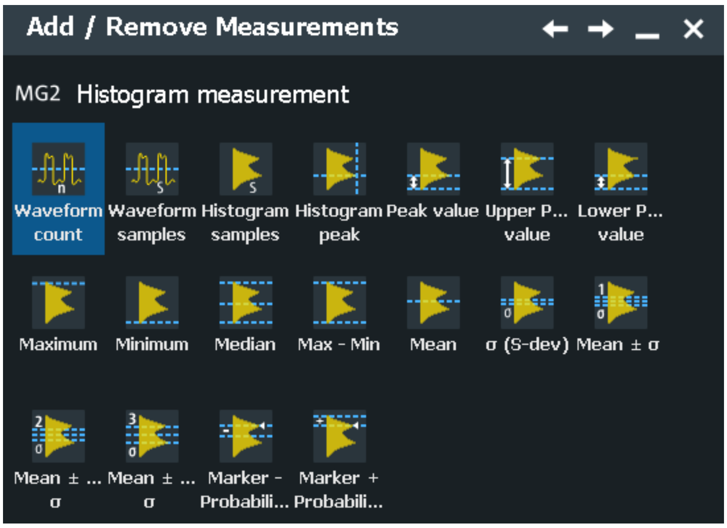
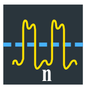
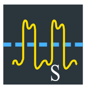
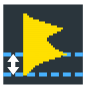
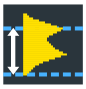
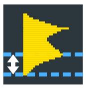
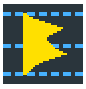
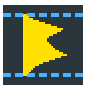
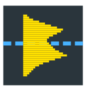
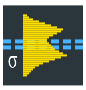
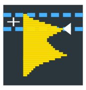
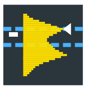


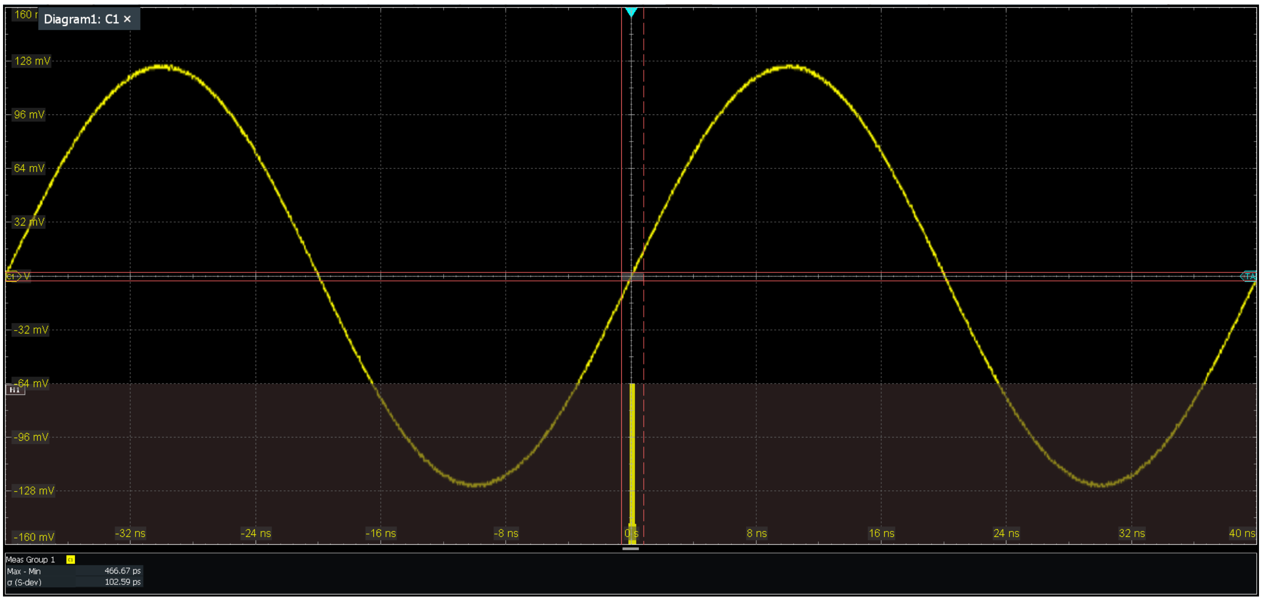
Table 8-8: Histogram measurements

	Meas. type	Description/Result
	Waveform count	The number of acquisitions (waveforms) the histogram is based on
	Waveform samples	The number of samples from the most recent acquisition included in the current histogram
	Histogram samples	The number of samples from all acquisitions included in the current histogram
	Histogram peak	The maximum count value in the histogram
	Peak value	The signal value at the histogram peak
	Upper peak value	The signal value at the maximum count value in the upper half of the histogram
	Lower peak value	The signal value at the maximum count value in the lower half of the histogram
	Maximum	The highest signal value with a probability > 0
	Minimum	The lowest signal value with a probability > 0
	Median	The signal value for which half the samples lie above, the other half below in the histogram The sample count of one signal value after the other are accumulated until half the total number of samples in the histogram is reached. The signal value for which 50% of the samples are accumulated is the median.
	Max - Min	The range of signal values with a probability > 0
	Mean	The weighted arithmetic average of the histogram
	σ (S-dev)	Standard deviation of the sample numbers

	Meas. type	Description/Result
	Mean ±σ	The range between (mean value + standard deviation) and (mean value - standard deviation)
	Mean ±2*σ	The range between (mean value + 3 * standard deviation) and (mean value - 2 * standard deviation)
	Mean ±3*σ	The range between (mean value + 3 * standard deviation) and (mean value - 2 * standard deviation)
	Marker + Probability %	The marker value (according to the selected probability domain marker type) plus the defined limit. Note that the value is restricted to the histogram range.
	Marker - Probability %	The marker value (according to the selected probability domain marker type) minus the defined limit. Note that the value is restricted to the histogram range.

Rough jitter evaluation using a histogram

You can use a horizontal histogram to perform a rough jitter measurement. Define a histogram for a narrow amplitude range close to the trigger time. The "Max-Min" value indicates the peak jitter, while the "StdDev" value indicates the RMS jitter.



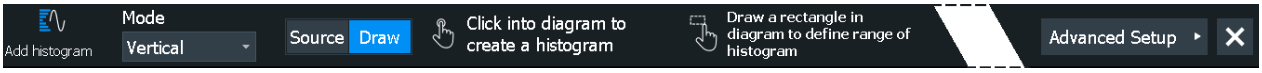
In addition to histograms on channel, math and reference waveforms, histograms can be created based on statistic measurement results. These histograms are enabled in the "Result Analysis" tab, see Chapter 8.2.11, "Result analysis", on page 356.

8.2.8.3 Creating histograms

Histograms can be used to evaluate the sample value occurrences directly. They are a prerequisite for histogram measurements.

To create a histogram quickly with toolbar icons

1. Select the waveform for which you need a histogram.
2. Tap the histogram icon on the toolbar.
3. Select the histogram type on the toolbar assist: "Mode" = vertical for an amplitude histogram, or horizontal for a time-based histogram.



4. Tap the diagram with the waveform to be measured, or draw a rectangle on the screen to define the area for histogram calculation.
   Alternatively, tap "Source" and select the waveform.

   The histogram range is indicated in the diagram and a histogram with the selected waveform as a source is defined and displayed.
5. To adjust the histogram range:
   a) Double-tap the histogram.
   b) Tap "Advanced Settings" on the toolbar assist.
   c) Adjust the start and stop values.

To create and configure a histogram in the dialog box

1. Select "Menu" > "Apps" > "Analysis" tab > "Histogram".

   The "Histogram" dialog box is displayed.

2. To create a histogram, tap the "Add" icon in the upper right corner of the dialog box.

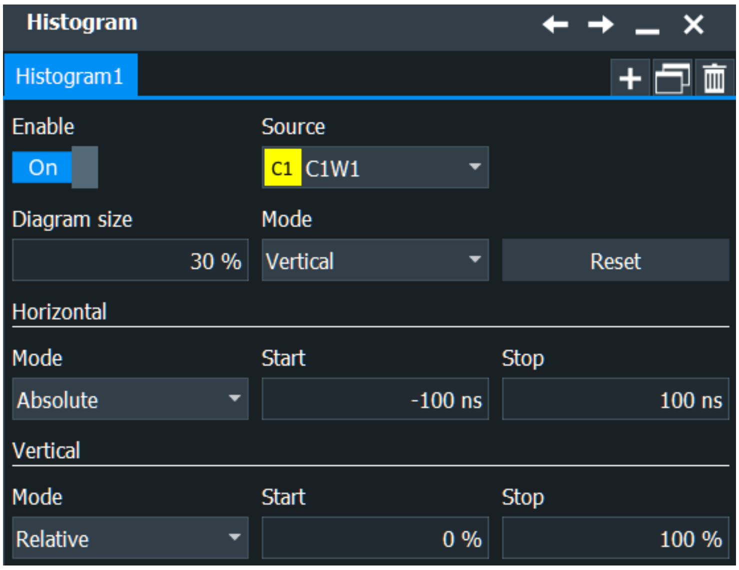
3. To copy an existing histogram and configure a new one based on those settings, tap the "Copy" icon.
4. To change the name of a histogram, double-tap the tab label. Enter a name for the histogram using the on-screen keyboard.
5. Select a "Source" for the histogram. The source can be any input signal, math or reference waveform.
6. Define the histogram "Mode": vertical for an amplitude, horizontal for a time-based histogram.
7. Define the range of the waveform for which the histogram is to be generated. Enter the start value and the stop value in x and in y direction, either as absolute or relative values.

8. Enable the histogram.

8.2.8.4 Histogram setup

"Menu" > "Apps" > "Analysis" tab > "Histogram"

In this dialog box, you configure histograms on which you can perform further measurements.



Enable
Enables or disables the histogram evaluation and display. The histogram settings are kept until the histogram is deleted.

Source
Defines the source of the histogram. Any analog channel waveform, math or reference waveform can be selected. Also measurements can serve as histogram source. In this case, the density distribution of the results of the main measurement is displayed.

Remote command:
`LAYout:HISTogram:SOURce` on page 1521

Diagram size
Defines the size of the histogram in percent of the diagram.

Mode
Defines the type of histogram.

"Vertical"	Amplitude histogram (horizontal bars across amplitude)
"Horizontal"	Time histogram (vertical bars over time). For spectrum waveforms, horizontal histograms over spectrum are not available.

Remote command:
`LAYout:HISTogram:MODE` on page 1521

Reset

Resets the values to begin a new histogram.

Remote command:
LAYout:HISTogram:RESet on page 1525

Mode (Horizontal / Vertical)

Defines whether the value range limits are entered as absolute or relative values.

Remote command:
LAYout:HISTogram:HORZ:MODE on page 1522
LAYout:HISTogram:VERTical:MODE on page 1523

Start, Stop (Horizontal)

Defines the horizontal value range of the histogram.

Remote command:
LAYout:HISTogram:HORZ:ABSolute:STARt on page 1522
LAYout:HISTogram:HORZ:ABSolute:STOP on page 1522
LAYout:HISTogram:HORZ:RELative:STARt on page 1523
LAYout:HISTogram:HORZ:RELative:STOP on page 1523

Start, Stop (Vertical)

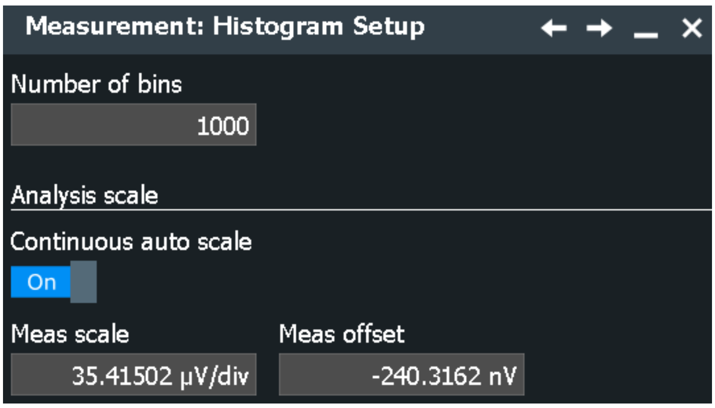
Defines the vertical value range of the histogram.

Remote command:
LAYout:HISTogram:VERTical:ABSolute:STARt on page 1523
LAYout:HISTogram:VERTical:ABSolute:STOP on page 1524
LAYout:HISTogram:VERTical:RELative:STARt on page 1524
LAYout:HISTogram:VERTical:RELative:STOP on page 1524

8.2.8.5 Settings for histogram measurement

Access: [Meas] > "Meas Group" tab > "Histogram" category

You can perform measurements on histograms. Before, you have to define a histogram, see Chapter 8.2.8.3, "Creating histograms", on page 347.



The measurement parameters are common for all histograms measurements, you find them directly on the "Meas Group" tab.

Source

Selects the histogram on which the measurement is based. Before, you have to define a histogram, see Chapter 8.2.8.3, "Creating histograms", on page 347.

Remote command:
`MEASurement<m>:HISTogram:SELect` on page 1526

Probability domain marker, Reference

Defines the marker reference in the probability domain.

"Peak"	The y-value with the maximum sample value in the histogram
"Upper Peak"	The y-value at the maximum sample value in the upper half of the histogram
"Lower Peak"	The y-value at the maximum sample value in the lower half of the histogram
"Maximum"	The highest y-value with a probability > 0
"Minimum"	The lowest y-value with a probability > 0
"Median"	The y-value for which half the samples lie above, the other half below in the histogram.
"Mean"	The weighted arithmetic average of the histogram

Remote command:
`MEASurement<m>:HISTogram:PROBability:TYPE` on page 1527

Probability domain marker, Delta

Defines a range around the marker.

Remote command:
`MEASurement<m>:HISTogram:PROBability:LIMit` on page 1527

8.2.9 Jitter measurements

Jitter measurements are availabe if one of the following options is installed:

- R&S RTO-K12
- R&S RTO-K133
- R&S RTO-K134

See Chapter 18.1, "Jitter measurements (Options R&S RTO-K12/-K13)", on page 1066.

8.2.10 Protocol measurements (option R&S RTO-K35)

Option R&S RTO-K35 provides specific measurements on automotive and Ethernet protocols.

8.2.10.1 Overview of protocol measurements

If option R&S RTO-K35 is installed, enhanced measurements on the supported serial buses are possible. With it, you can measure various dependencies between the frames.

Additional to option R&S RTO-K35 you need one of the following serial protocol options:

- I2C (R&S RTO-K1)
- SPI (R&S RTO-K1)
- UART/RS232 (R&S RTO-K2)
- CAN / CAN-FD (R&S RTO-K3/R&S RTO-K9)
- LIN (R&S RTO-K3)
- RFFE (R&S RTO-K40)
- SENT (R&S RTO-K10)
- Ethernet (100BASE-Tx) (R&S RTO-K8)
- 100BASE-T1 (R&S RTO-57)

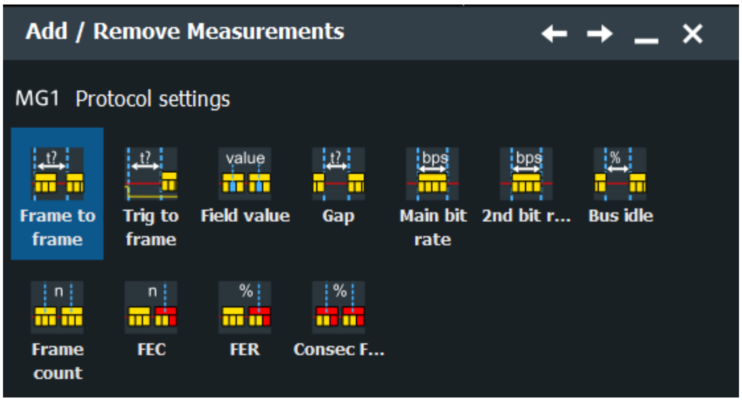


Table 8-9: Protocol measurements

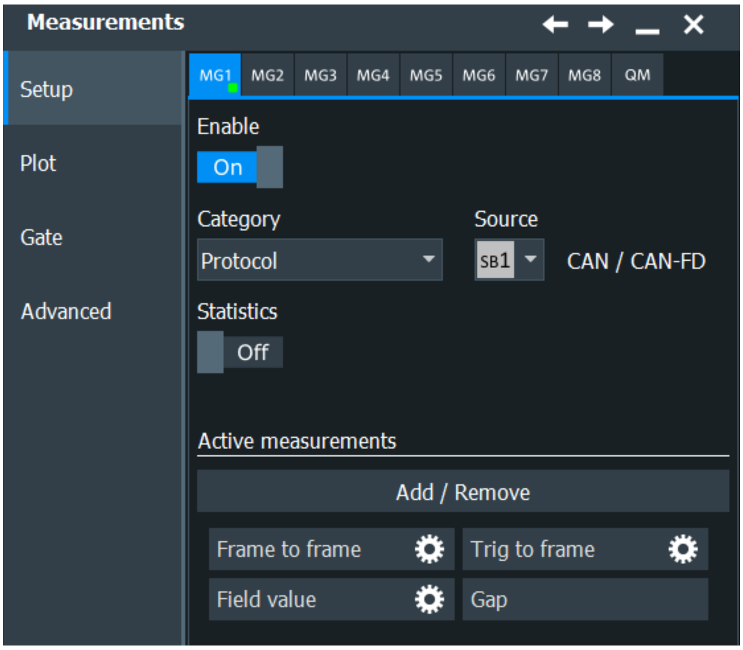
	Meas. type	Description / result
	Field value	Value of a field over time
	Frame to frame	The distance between two frames
	Trigger to frame	The distance between a defined frame and the next trigger signal
	Gap	Measures a gap, period at which the bus is idle. The distance of a gap can only be measured between two identified frames.
	Main bit rate	Bit rate as defined by the standard
	2nd bit rate	Additional bit rate, protocol dependent For example: "Data rate" for the CAN-FD protocol.
	Bus idle	Calculates the bus idle time
	Frame count	Number of all frames within the acquisition window
	Frame error count	Sum of all frames with errors within the acquisition window

	Meas. type	Description / result
	Frame error rate	Sum of all frames with errors divided by all frames within the acquisition window
	Consecutive frame error rate	Measures the rate at which at least two consecutive frames have an error

8.2.10.2 Settings for protocol measurements

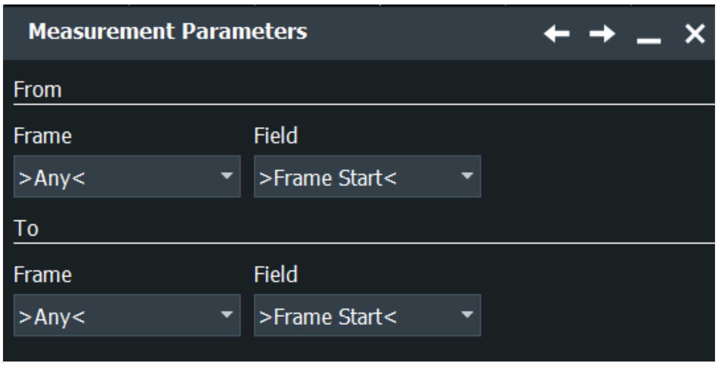
Access: [Meas] > "Meas Group" tab > "Protocol" category

Protocol measurements require that a supported protocol is enabled first.



If further settings are available for a measurement, a settings icon is shown beside the measurement's name in the "Active Measurements" list. Tap the icon to configure the measurement.

Frame to frame
Opens a dialog to set up the parameters for the frame to frame measurement. You can define a start frame ("From") and an end frame ("To") for the measurement.



The types of available frames and fields are depending on the enabled protocol.

Table 8-10: Available protocol frames

Protocol	Frame	Field
I2C	W R	Address
		2. Addr
		ACK
		R/W
		Data
SPI	MISO MOSI	Word
CAN/CAN FD	Remote Data	ID FDF BRS ESI DLC SCV P SC Data CRC ACK
	Overload	-
LIN	WAKE	-
	Data	ID Data

Protocol	Frame	Field
RFFE	Register 0 Write Register Write Register Read Extended Register Write Extended Register Read Extended Register Write Long Extended Register Read Long Interrupt Summary and Identification Masked Write Master Ownership Handover Master Write Master Read Master Context Transfer Write Master Context Transfer Read	SA MID BC Addr Mask Data
Ethernet	MAC Sleep EOS	Preamble SFD Dest Src Len/Type Data FCS
	Idle	Triplet
100BASE-T1	MAC Fill	Preamble SFD Dest Src Len/Type Data FCS
	Idle	Triplet

Also, if a label list is loaded and enabled for the protocol, the symbolic label names can be selected from the "Frame"/"Field" value list.

"Frame" Selects the type of start frame (from) / end frame (to).

"Field" Selects the type of start field (from) / end field (to).

"Value" Sets a value for the selected field.

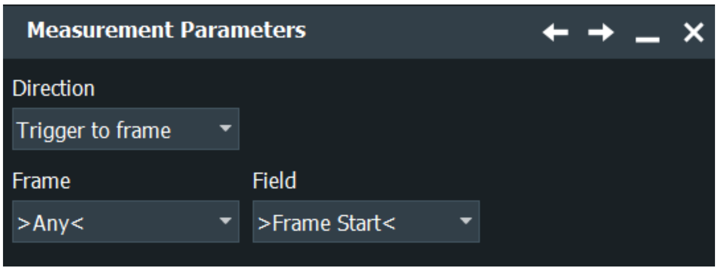
Remote command:
MEASurement<m>:PROTocol:F2FRame:FRMFrom on page 1539
MEASurement<m>:PROTocol:F2FRame:FRMTo on page 1539
MEASurement<m>:PROTocol:F2FRame:FLDFrom on page 1538
MEASurement<m>:PROTocol:F2FRame:FLDTo on page 1538

MEASurement<m>:PROTocol:F2FRame:VALFrom on page 1539
MEASurement<m>:PROTocol:F2FRame:VALTo on page 1539

Trigger to frame

Opens a dialog to set up the parameters for the trigger to frame measurement.

The types of available frames are depending on the enabled protocol, see Table 8-10.



"Direction" Selects the direction for the measurement, from the trigger to the next frame (start),

"Frame" Selects the type of frame.

"Field" Selects the type of field.

"Value" Sets a value for the selected field.

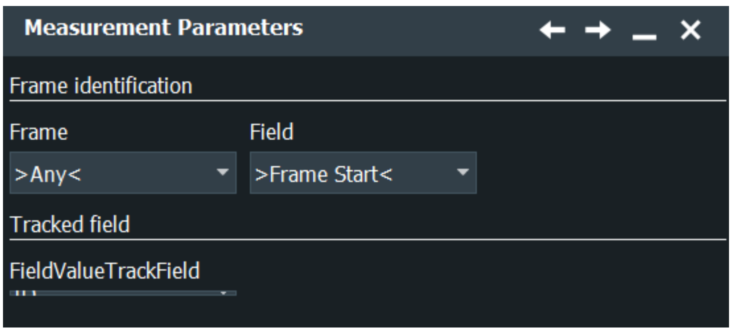
Remote command:
MEASurement<m>:PROTocol:T2FRame:DIRection on page 1542
MEASurement<m>:PROTocol:T2FRame:FLD on page 1542
MEASurement<m>:PROTocol:T2FRame:FRM on page 1542
MEASurement<m>:PROTocol:T2FRame:VALue on page 1543

Field value

Opens a dialog to set up the parameters for the field value measurement. You can first identify a frame with specific field value and then track a selected field of this frame.

The types of available frames and fields depend on the enabled protocol, see Table 8-10.

If a label list is loaded and enabled for the protocol, sometimes there are defined calculations and formatting for the displayed field value. In this case, the measurement displays the value according to the definition in the label list.



"Frame" Selects the type of frame.

"Field" Selects the type of field.

"Value" Sets a value for the selected field.

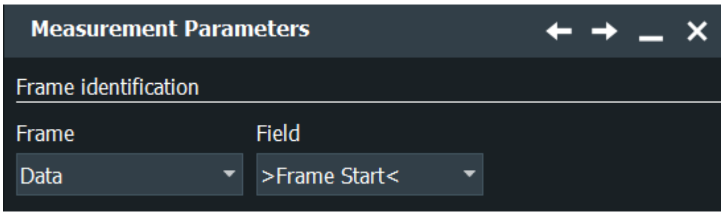
"Field to be tracked" Selects the type of field which value is tracked over time.

Remote command:
MEASurement<m>:PROTocol:FLDValue:FLD on page 1540
MEASurement<m>:PROTocol:FLDValue:FRM on page 1540
MEASurement<m>:PROTocol:FLDValue:TRCK on page 1540
MEASurement<m>:PROTocol:FLDValue:VAL on page 1540

Main bit rate / 2nd bit rate
Opens a dialog to set up the parameters for the main bit rate/ 2nd bit rate measurement. You can identify a frame and a specific field value.

The types of available frames and fields depend on the enabled protocol, see Table 8-10.

If a label list is loaded and enabled for the protocol, sometimes there are defined calculations and formatting for the displayed field value. In this case, the measurement displays the value according to the definition in the label list.



"Frame" Selects the type of frame.

"Field" Selects the type of field.

"Value" Sets a value for the selected field.

Remote command:
MEASurement<m>:PROTocol:MBITrate:FLD on page 1541
MEASurement<m>:PROTocol:MBITrate:FRM on page 1541
MEASurement<m>:PROTocol:MBITrate:VAL on page 1541
MEASurement<m>:PROTocol:SBITrate:FLD on page 1541
MEASurement<m>:PROTocol:SBITrate:FRM on page 1541
MEASurement<m>:PROTocol:SBITrate:VAL on page 1542

8.2.11 Result analysis

The behavior of measurement results over time can be evaluated in different ways:

- Statistics
- Long-term measurements
- Histograms on measurement results
- Track

8.2.11.1 Statistics

Statistics can be compiled for each measurement group separately, and also for long-term measurements. If enabled, statistical results are shown in the result box.

By default, all events in each acquisition are measured and incuded in the statitics: the measurement result is not only determined once within one acquisition, but repeatedly, if available. More results provide a larger basis for statistical evaluation.

To obtain precise results, additional measurement settings can be useful:

- Reference/signal levels: configuring user-defined levels can compensate for irregular data, see Chapter 8.2.4, "Reference levels", on page 311.
- Gate areas: restricting the waveform range for measurement can eliminate irregular data, see Chapter 8.2.3, "Measurement gates", on page 308.
- Defining a "Signal threshold" for time, area and counting measurements can eliminate noise from the evaluation, see "Signal threshold" on page 327.
- Spectrum measurements: you can eliminate noise from the evaluation, see Threshold and "Peak excursion" on page 298

► To enable statistics, use one of the following ways.
Make sure to select the subtab of the measurement group for which you want to compile statistics.
  - On the toolbar assist, enable "Statistics".
  - On the "Meas Group" tab, enable "Statistics".

If statistics are enabled, the following results are calculated. You can select which results are shown in the result table, see Chapter 8.2.2.3, "Display settings for results", on page 307.

Label	Description
+Peak	Positive peak value (maximum)
-Peak	Negative peak value (minimum)
μ (Avg)	Average
RMS	Root mean square
σ (S-dev)	Standard deviation
Event count	Number of measured events (e.g. rising edges, pulses etc.)
Wave count	Number of waveforms (acquisitions) the measurement is based on

Remote commands:

- MEASurement<m>:RESult[:ACTual]? on page 1488
- MEASurement<m>:RESult:AVG? on page 1488
- MEASurement<m>:RESult:COUNt? on page 1490
- MEASurement<m>:RESult:EVTCount? on page 1488
- MEASurement<m>:RESult:NPEak? on page 1488

- MEASurement<m>:RESult:PPEak? on page 1488
- MEASurement<m>:RESult:RMS? on page 1488
- MEASurement<m>:RESult:STDDev? on page 1489
- MEASurement<m>:RESult:WFMCount? on page 1489
- MEASurement<m>:RESult:STARt? on page 1490
- MEASurement<m>:RESult:STOP? on page 1490
- MEASurement<m>:ARNames on page 1488
- MEASurement<m>:ARES? on page 1487

The peak and average values and the standard deviation of the long-term points are also shown in the graph of the long-term measurement.

Stopping and starting the acquisition does not reset statistics but only stops and continues them.

The instrument only resets statistical evaluation if you change measurement setup:

- Select measurement
- Create or modify gate
- Enable/disable long-term measurement and histogram
- Enable continuous autoscale with enabled histogram
- Switch on/off channels
- Enable/disable cursors
- Tap "Reset" or "Clear sreen results"

After a reset, new statistics are compiled beginning with the next acquired waveform.

If limit and margin check is enabled, the icons in the result table indicate if statistical results exceed a limit or margin. These violations do not initiate an action.

8.2.11.2 Long-term measurements

Long-term measurements show the behavior of measurement results over a longer time or for many samples. Therefore the measurement results of a specified time period are summarized into one long-term point. For each point, the current value measured at the end of the time period is written to the long-term waveform. In addition, statistical results for each time period are calculated, saved, and reset. This reset avoids constantly rising maximum or constantly falling minimum values until the end of the measurement.

You can define the number of long-term points and export the long-term data, including statistical results. The measurement histogram is a vertical histogram shown in the long-term diagram.

Long-term measurements are done on the selected "Measurement for analysis / math".

See also: "Enable (Long term)" on page 361.

Performing long-term measurements

1. On the "Meas" menu, select "Result Analysis".
2. Select the subtab for the measurement group you want to configure.
3. Under "Long term", tap "Enable".
4. Since the waveform can change in the process of time, enable "Continuous auto scale" to adapt the scaling automatically.
   Alternatively, tap the "Auto scale" to adjust the scale once and to see the long-term waveform.
5. Tap "Horizontal scaling".
6. Define the "Number of points" to be shown in the long-term diagram.
7. Set the "Scale mode" that defines the period of time from which one long-term point is created.
   See "Scale mode" on page 364 for setting details.

   For each long-term measurement point, the current measurement value is added to the long-term waveform.
8. If you need the statistical data of the long-term points:
   a) Tap "Result Analysis" to return to the measurement settings.
   b) Enable statistics.
   c) Let the measurement run and export the data when finished.

8.2.11.3 Histograms on measurement results

Histograms are available not only for channel, math and reference waveforms, but also on measurement results. These histograms cannot be configured, and they are shown in a separate diagram. The source is a measurement, and all events in all acquisitions are are measured and included in the histogram. If the histogram is based on long-term measurement, it is shown in the long-term diagram.

The histogram is built from the results of the selected "Measurement for analysis / math".

See also: "Enable (Histogram)" on page 361.

8.2.11.4 Track

The track is a waveform that shows measurement values in time-correlation to the measured signal. It is the graphical interpretation of all measurement values of a single acquisition.

The track is available for most amplitude/time measurements (except for High, Low, Amplitude, Max, Min, Peak to peak, Mean, RMS, S-dev, Pos. and Neg. overshoot, and Area), and for jitter measurements.

Enabling the track enables also the Continuous auto scale and Measure all events in each acquisition.

To analyze the track, you can use cursor measurements and zoom.

Creating a track waveform

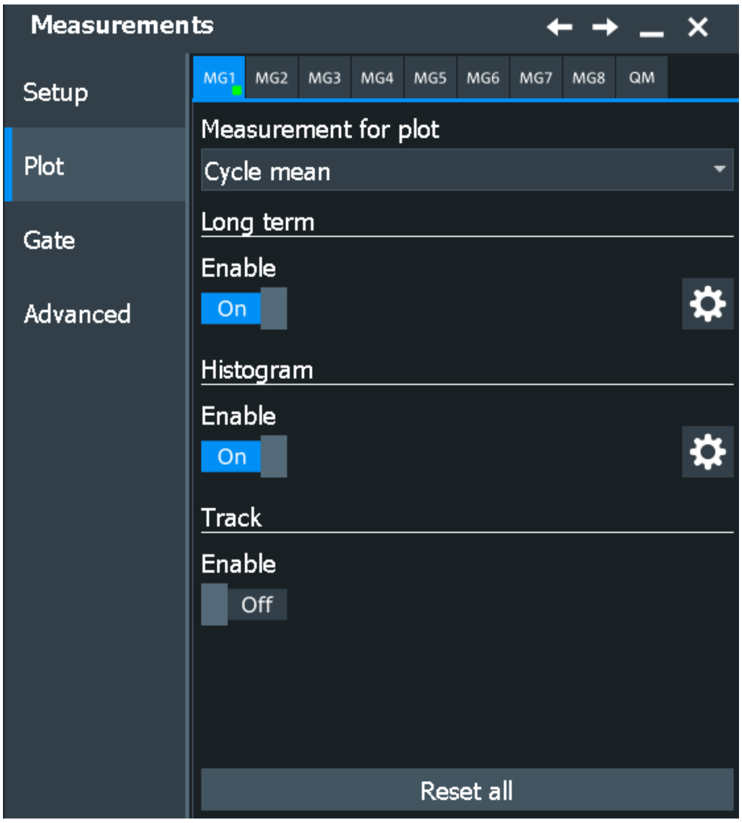
1. Configure the measurement group.
2. Activate the measurement group.
3. Select the "Result Analysis" tab.
4. If several measurements are enabled, select the "Measurement for analysis / math".
5. Enable the track.

See also: "Enable (Track)" on page 362.

8.2.11.5 Settings for result analysis

The settings in the "Result Analysis" tab activate and configure long-term measurements, statistical calculations, and the histogram of measurement results over a longer period of time.

For scaling settings of the long-term diagram, see Chapter 8.2.11.6, "Horizontal long-term scaling", on page 363.



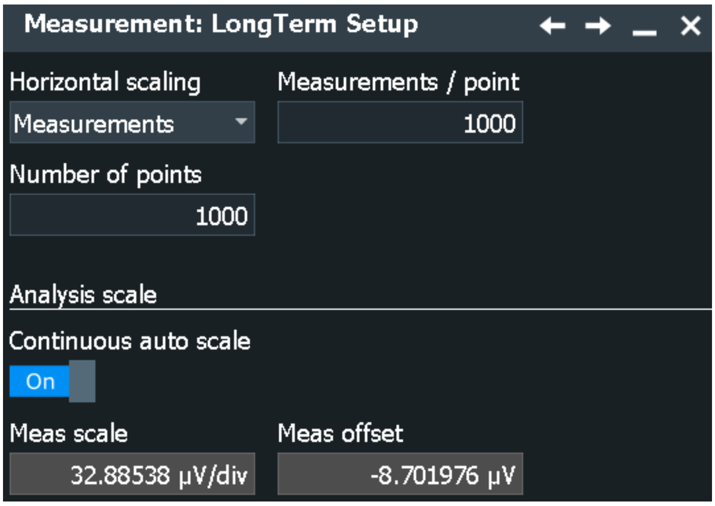
Measurement for plot
Selects the measurement that is used as source for mathematic calculations, long-term measurements, and histograms. These evaluations are performed on one measurement only, not on all measurements of the group. Statistics are calculated for all measurements.

The selected measurement cannot be disabled in the "Add / Remove Measurement" dialog box.

Remote command:
MEASurement<m>:MAIN on page 1485

Enable (Long term)
Enables long-term measurement of the main measurement.

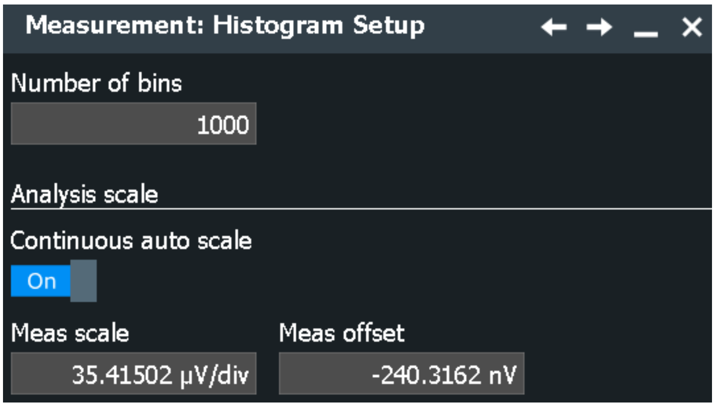
Long-term measurements are performed on the "Measurement for plot".



Remote command:
MEASurement<m>:LTMeas[:STATe] on page 1536

Enable (Histogram)
Displays a histogram of measurement results - the cumulative occurrence distribution of measurement results in a graphic. Enabling the histogram enables also the statistics and "Measure all events in each acquisition".

Long-term measurements are performed on the "Measurement for plot".



Remote command:
MEASurement<m>:STATistics:HISTogram on page 1532

Number of bins

Sets the number of bins - the number of vertical bars that build the histogram.

If "Continuous auto scale" is enabled, the instrument determines the number of bins based on the timebase, the current measurements, and other settings. To set the number of bins manually, disable "Continuous auto scale".

Remote command:
MEASurement<m>:STATistics:HBINs on page 1533

Enable (Track)

Enables the track of measurement results over time and displays the track waveform. It is the graphical interpretation of all measurement values of a single acquisition.

The track is available for most amplitude/time measurements (except for High, Low, Amplitude, Max, Min, Peak to peak, Mean, RMS, S-dev, Pos. and Neg. overshoot, and Area), and for jitter measurements.

Enabling the track enables also the Continuous auto scale and Measure all events in each acquisition.

Before you can enable the track, activate the appropriate measurement.

If option R&S RTO-K12 basic jitter analysis is installed, you can use tracks to display the jitter measurement results as a time-correlated waveform, see Chapter 18.1.4, "Track of jitter measurement results", on page 1075.

With option R&S RTO-K5 I²S audio signals, you can use the track for protocol measurements on decoded audio buses, see Chapter 13.8.5, "Track", on page 596.

Remote command:
MEASurement<m>:TRACk[:STATe] on page 1543

Reset

Immediately resets the histogram, the long-term measurement and the statistics.

Stopping and starting the acquisition does not reset these analyses but only stops and continues them.

To delete all results, waveforms and history, select "Display" menu > "Clear all".

Remote command:
MEASurement<m>:STATistics:RESet on page 1534

Analysis scale
The measurement scale of a long-term measurement diagram or measurement histogram can be set automatically by the instrument, or manually.

Use automatic scaling if the measurement is running and you cannot see the expected results.

"Continuous auto scale"
Performs an automatic scaling whenever the long-term waveform or the histogram does not fit in the diagram during the measurement period.

"Vertical" Defines the vertical scaling per division for long-term measurement period and the measurement histogram.

"Offset" Defines an offset for the long-term measurement and the measurement histogram.

Remote command:
MEASurement<m>:VERTical:CONT on page 1535
MEASurement<m>:VERTical:AUTO on page 1535
MEASurement<m>:VERTical:SCALe on page 1536
MEASurement<m>:VERTical:OFFSet on page 1535

Measure all events in each acquisition
Normally, only one measurement is performed for each acquired waveform to get best performance. If "Measure all events in each acquisition" is enabled, more than one result is taken from one acquired waveform and the results are included evaluation. For example, the rise time is measured on all pulses in the waveform, not only on the first.

The result box shows only the first result of the waveform, the following results are used only for evaluation.

All event results are also considered in limit and margin checks and can initiate an action. However, the icons in the result box only indicate violations of the first result.

Measuring all events is enabled automatically when calculating statistics and histograms. It can also be useful when generating tracks; however, it reduces the performance of the instrument.

The number of considered results can be restricted: "Limit" sets the maximum number of measurement results per acquisition.

In firmware versions < 4.00, this setting was named "Multiple measurement".

Remote command:
MEASurement<m>:MULTiple on page 1532
MEASurement<m>:MNOMeas on page 1532

8.2.11.6 Horizontal long-term scaling

In this dialog box, you define the horizontal scale of long-term measurement diagrams. The length of the long-term measurement is defined by the number of points.

If option R&S RTO-K5 I²S Audio Signals is installed, the trend diagram is configured here.

Number of points
Defines the total number of points to be displayed in the long-term measurement diagram.

Remote command:
MEASurement<m>:LTMeas:COUNt on page 1536

Scale mode
Defines when the points of a long-term measurement are created.

If statistics are enabled, each long-term measurement point shows the statistical mean and standard deviation of the results measured during the defined period.

If statistics are disabled, the first measurement result of each period is taken as long-term measurement point.

"Time"	Sets one long-term measurement point for the time defined in "Time/point".
"Waveforms"	Sets one long-term measurement point for several acquired waveforms. The number is defined in "Waveforms/point".
"Measurements"	Sets one long-term measurement point for several measurement results. The number is defined in "Measurements/point".

Remote command:
MEASurement<m>:STATistics:MODE on page 1533

Time / point
Defines the time to create one point of the long-term measurement. The "Time / point" value is a lower time limit. The actual time between two points depends on the acquisition and postprocessing time.

The long-term measurement is not a data logger with equidistant points as the time between two points varies.

This setting is only available if "Scale mode" is set to "Time".

Remote command:
MEASurement<m>:STATistics:RTIMe on page 1534

Measurement time
Defines the total duration of the long-term measurement: *Time/point * Number of points*.

This setting is only available if "Scale mode" is set to "Time".

Remote command:
MEASurement<m>:LTMeas:TIME on page 1537

Wfms / point
Defines the number of measured waveforms from which one point of the long-term measurement is created.

This setting is only available if "Scale mode" is set to "Waveforms".

Remote command:
MEASurement<m>:STATistics:RCOunt on page 1533

Measurements / point
Defines the number of measurement results from which one point of the long-term measurement is created.

This setting is only available if "Scale mode" is set to "Measurements".

Remote command:
MEASurement<m>:STATistics:RMEascount on page 1534

8.2.12 Limit and margin checks

Limit and margin checks evaluate whether the measurement result exceeds a specified value. Violations are indicated by icons in the result box. Furthermore, you can define actions that are performed on limit or margin violation, like saving the waveform or measurement results.

The following results are considered in limit and margin checks:

- All selected measurements.
- All measured events in an acquisition: all results can initiate an action. However, the icons in the result box indicate only violations of the first result.
- Statistical results. Limit and margin violations of statistical results are indicated by icons in the result box. These violations do not initiate an action.

See also: Chapter 8.2.2.1, "Measurement status", on page 306

8.2.12.1 Performing limit checks

1. Open the menu and select "Measure".
2. Select the "Advanced" tab.
3. Select the subtab of the measurement group you want to configure.
4. Under "Type", select "Limit only" to distinguish only between valid and non-valid values.
   Select "Margin&Limit" to check two values, where the margin is still valid, while the limit is not.
5. Tap "Setup". Define the valid value range for each active measurement.
   The margins must always be within the valid value range. If necessary, the limit or margin values are adapted to match the selected valid range.
   See also "Upper limit, Lower limit, Upper margin, Lower margin, Valid range" on page 366
6. Define what happens when the defined limits and margins are exceeded.

   For each action, define when the instrument starts it:
   - If the limits or margins are exceeded.

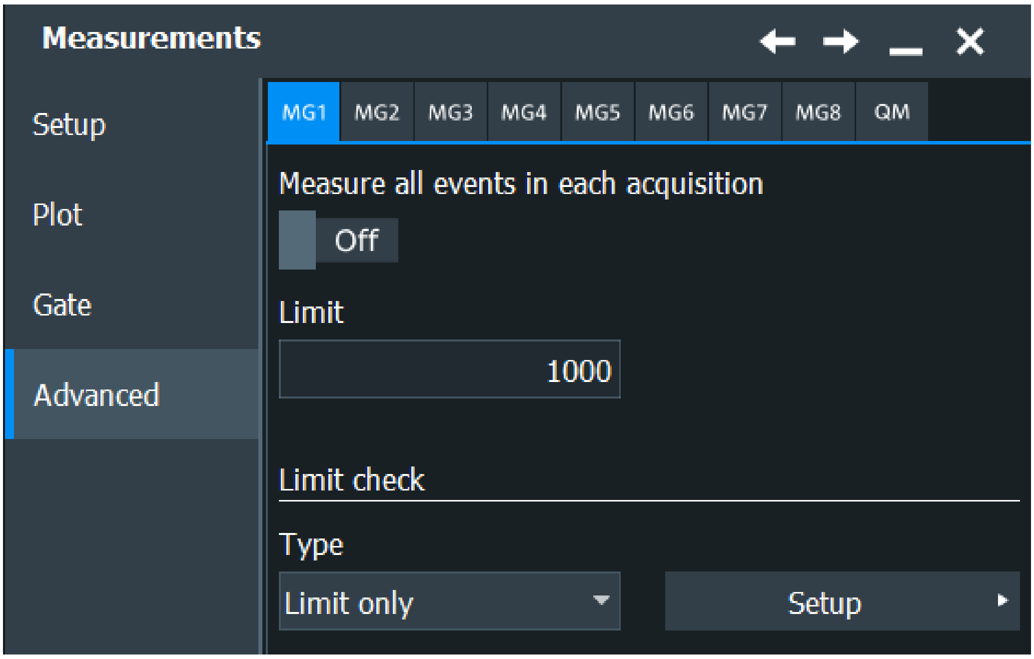
- If the measurement is completed without limit violations.
- Not at all.

As a result of the limit check, the specified actions are performed and an icon indicates the status in the result box.

8.2.12.2 Limit and margin settings

Access: [Meas] > "Advanced" tab

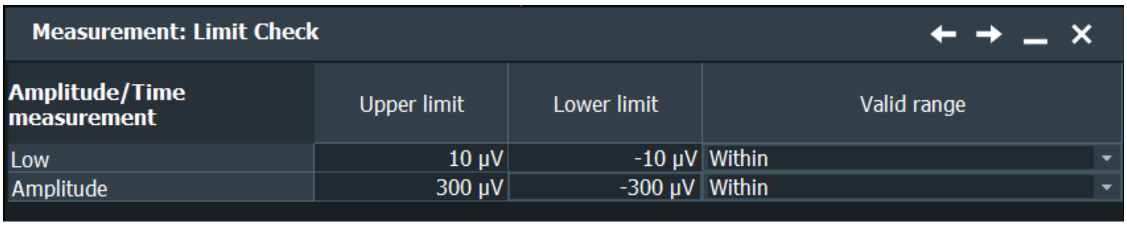
If a check is selected, tap "Setup" to set the limit and margin values and the range of valid measurement results



Type
Selects and enables the limit or margin check.

"Off"	No limit check is performed.
"Limit only"	Limits are checked for violation.
"Margin & Limit"	Margins and limits are checked for violation.

Upper limit, Lower limit, Upper margin, Lower margin, Valid range
Set the limits and margins for each measurement, and also specify the valid range.



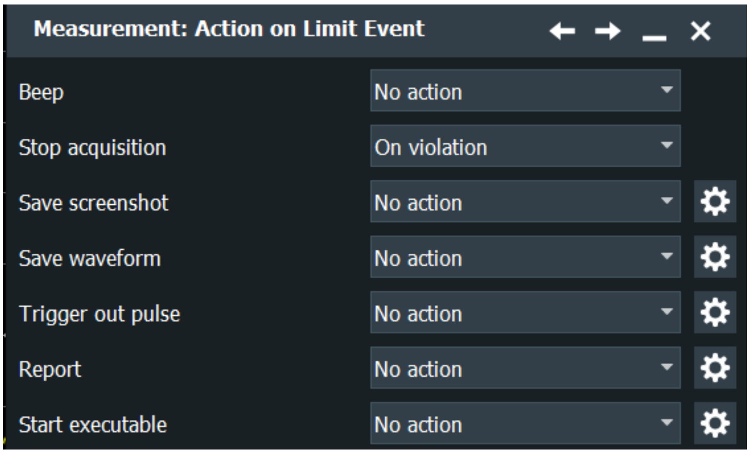
Limits are stricter than the margins for the value check. Thus, the margins must be within the valid range. If necessary, the limit and margin values are adapted according to the selected valid range.

The settings are only visible if "Limit check" is enabled.

Remote command:
MEASurement<m>:AMPTime:LCHeck<n>:LOWer:LIMit on page 1500
MEASurement<m>:AMPTime:LCHeck<n>:LOWer:MARGin on page 1501
MEASurement<m>:AMPTime:LCHeck<n>:UPPer:LIMit on page 1500
MEASurement<m>:AMPTime:LCHeck<n>:UPPer:MARGin on page 1501
MEASurement<m>:AMPTime:LCHeck<n>:VALid on page 1500
To check limits and margins of jitter measurements, use the AMPTime remote commands.
MEASurement<m>:EYEJitter:LCHeck<n>:LOWer:LIMit on page 1511
MEASurement<m>:EYEJitter:LCHeck<n>:LOWer:MARGin on page 1512
MEASurement<m>:EYEJitter:LCHeck<n>:UPPer:LIMit on page 1511
MEASurement<m>:EYEJitter:LCHeck<n>:UPPer:MARGin on page 1512
MEASurement<m>:EYEJitter:LCHeck<n>:VALid on page 1511
MEASurement<m>:HISTogram:LCHeck<n>:LOWer:LIMit on page 1528
MEASurement<m>:HISTogram:LCHeck<n>:LOWer:MARGin on page 1529
MEASurement<m>:HISTogram:LCHeck<n>:UPPer:LIMit on page 1528
MEASurement<m>:HISTogram:LCHeck<n>:UPPer:MARGin on page 1529
MEASurement<m>:HISTogram:LCHeck<n>:VALid on page 1528
MEASurement<m>:SPECtrum:LCHeck<n>:LOWer:LIMit on page 1518
MEASurement<m>:SPECtrum:LCHeck<n>:LOWer:MARGin on page 1519
MEASurement<m>:SPECtrum:LCHeck<n>:UPPer:LIMit on page 1518
MEASurement<m>:SPECtrum:LCHeck<n>:UPPer:MARGin on page 1519
MEASurement<m>:SPECtrum:LCHeck<n>:VALid on page 1518

8.2.12.3 Actions on limit check results

To define what happens when the limits and margins are exceeded, tap Action on limit event below the limit and margin ranges.



Actions are initiated by all measurements and all results measured on one acquisition.

Note that the violation actions do not distinguish between a margin violation and a limit violation. However, different icons are displayed in the result box.

For each action, you can define the event on which the action is initiated:

- On violation
  The action is initiated when the limits or margins are exceeded during the measurement.
- On successful completion
  The action is initiated when a defined number of acquisitions has been captured, and the limits or margins were not exceeded.

Independent of these actions, an icon is displayed in the result box, see Chapter 8.2.2, "Measurement results", on page 305.

Beep
Generates a beep sound.

Remote command:
`MEASurement<m>:ONViolation:BEEP` on page 1548

Stop acq
Stops data acquisition on violation.

Remote command:
`MEASurement<m>:ONViolation:ACQStop` on page 1548

Save Wfm
Saves the waveform data to the file.

Remote command:
`MEASurement<m>:ONViolation:WFMSave` on page 1549

Trigger Out Pulse
Creates a pulse on the Trigger Out connector on limit violation.

When "Trigger Out Pulse" is used, the trigger control option "Enable trigger out" is disabled. Thus, the trigger-out pulse is created only on limit violation but not when a trigger occurs. The pulse is provided always with the minimum delay of 800 ns, the "Delay" cannot be set.

See also: "Trigger out signal setup" on page 228.

Remote command:
`MEASurement<m>:ONViolation:TRIGgerout` on page 1549

Report
Creates and saves a report using the settings defined in "File" menu > "Report Setup".

Remote command:
`MEASurement<m>:ONViolation:REPort` on page 1549

Start Executable
Starts an external application. Tap "Config Executable" to set the application path and parameters.

Remote command:
MEASurement<m>:ONViolation:RUNexec on page 1549

8.3 Quick measurements

Quick measurement performs a set of up to eight amplitude/time measurements on one source, simply by tapping the "Quick measurement" toolbar icon. The results are displayed in a results box. You can configure the measurement to be included in quick measurement. The current configuration can be saved to repeat the measurement quickly.

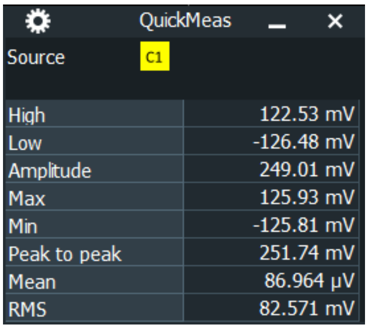
8.3.1 Starting the quick measurement

If the "Quick meas" icon is not visible on the toolbar, add it to the toolbar: see Chapter 3.3.6.2, "Configuring the toolbar", on page 57.

1. Tap the waveform that you want to measure.
2. Tap the "Quick measurement" icon on the toolbar.

3. Tap the diagram.

   The result box shows the results of the default quick measurement.



8.3.2 Configuring the quick measurement

The default configuration of the quick measurement includes already 8 amplitude measurements. If these measurements do not fit the measurement task, you can modify the selection.

1. Open "Menu" > "Measure" > "Meas Group".
2. Select the "QM" subtab.

3. Tap "Add/Remove Measurements".
4. Disable all measurements that you do not need.
5. Select the measurements that you want to add to the quick measurement.
6. Tap "OK".
7. A "Settings" icon indicates whether further settings are required. Tap the measurement. A dialog box opens, where you can adjust the measurement.
8. Tap "Set as QuickMeas" to save the setup.

   The saved iQuickMeas setup is used when you start a new quick measurement.
9. "Enable" the measurement.

Set as QuickMeas
Saves the current QuickMeas setup. The saved QuickMeas configuration is used when you start a new quick measurement. It remains until you save another QuickMeas setup, or until you reset the instrument to "Factory defaults". The "Set as QuickMeas" is not available if the current configuration already has been saved.

9 Spectrum analysis

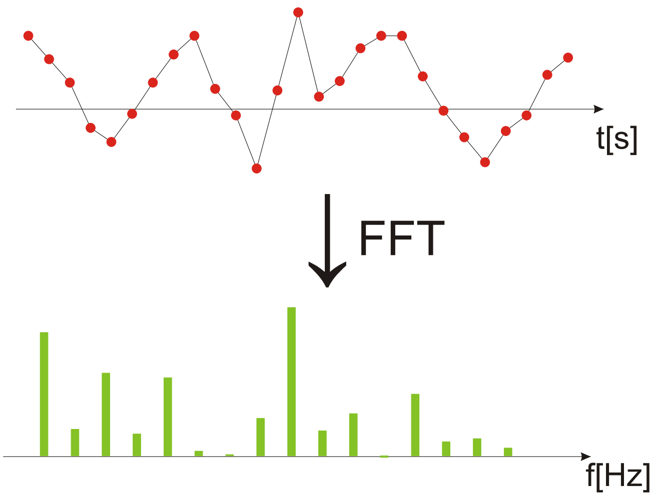
The R&S RTO provides two ways of spectrum analysis:

- Basic FFT calculation, which is included in the firmware
- Spectrum Analysis option R&S RTO-K18, which provides a wide range of analysis possibilities, for example, spectrogram, cursor and automatic measurements.

9.1 FFT analysis

9.1.1 Fundamentals of FFT analysis

During FFT analysis, a signal in the time domain is converted to a spectrum of frequencies. As a result, either the magnitude or the phase of the determined frequencies can be displayed. FFT analysis can be restricted to an extract of the original time base, and the results display can be restricted to a specified frequency range.

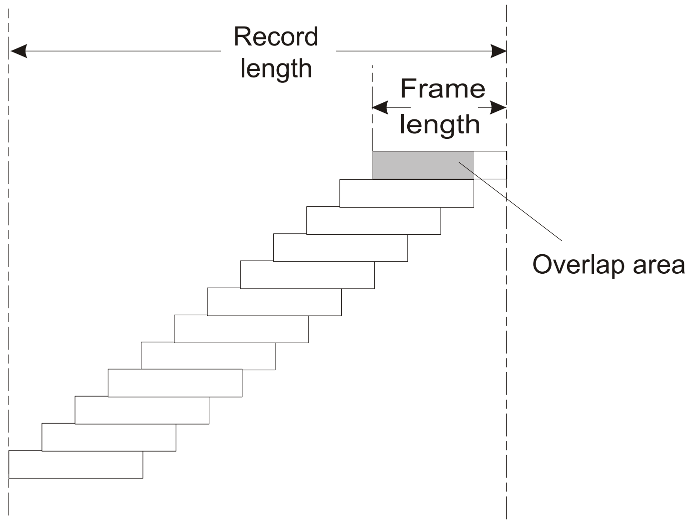


Frames/Segments

To convert the time domain signal to a frequency spectrum, an FFT (Fast Fourier Transformation) unit is used which converts a vector of input values into a discrete spectrum of frequencies.

Conventional oscilloscopes calculate one FFT per capture. The R&S RTO can calculate multiple FFTs per capture by dividing one capture into several *segments*, or *frames*. Thus, the R&S RTO can visualize how the frequency content of a signal changes over time which helps to detect intermittent or sporadic signal details. Furthermore, the R&S RTO allows consecutive frames to overlap. This is especially useful in conjunction with window functions since it enables a gap-free frequency analysis of the signal.

The overlapping factor can be set freely. The higher the overlap factor, the more frames are used. This leads to more individual results and improves detection of transient signal effects. However, it also extends the duration of the calculation. The size of the frame depends on the number of input signal values (record length), the overlap factor, and the FFT size (number of samples used for FFT calculation).



Window functions

Each frame is multiplied with a specific window function after sampling in the time domain. Windowing helps minimize the discontinuities at the end of the measured signal interval and thus reduces the effect of spectral leakage, increasing the frequency resolution.

There are several window functions that can be used in FFT analysis. Each of the window functions has specific characteristics, including some advantages and some tradeoffs. Consider these characteristics carefully to find the optimum solution for the measurement task.

For details, see "Window type" on page 382.

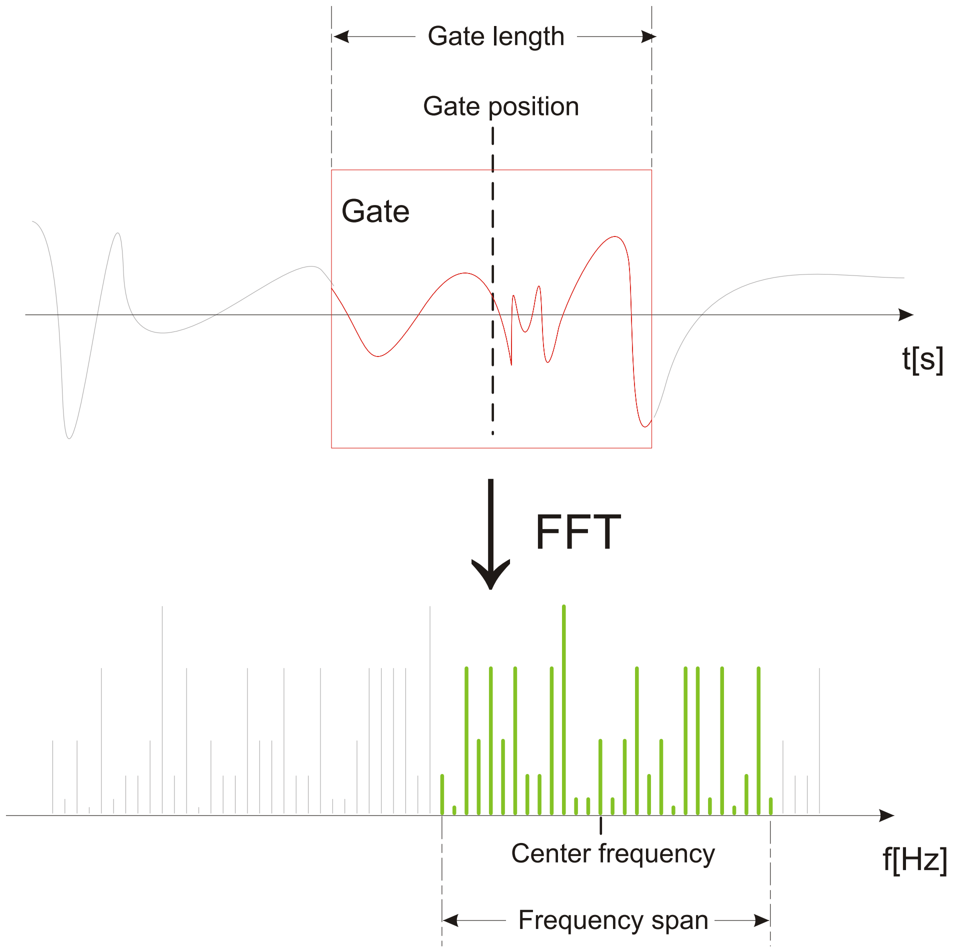
Combining FFT results

After the window function has been applied to the FFT results for each segment, the results for all segments of the data acquisition must be combined to obtain the final waveform. Various arithmetic functions are available for FFT segments, such as averaging, enveloping, or minimum and maximum calculation.

Gating functions

You can restrict the time base of the input signal for which FFT analysis is to be performed. There are various methods to do so:

- Define absolute start and stop times for the time base extract
- Define relative start and stop values that define a percentage of the original time base
- Couple the time base extract for FFT to an active zoom area.



Restricting the result range

You can restrict the results of the FFT analysis to a specified frequency range. The frequency range can be defined in two ways:

- Define a center frequency and frequency span
- Define start and stop frequencies

Using the new cursor functions for spectrum waveforms you can easily determine the results for the current center frequency by moving the cursor to that frequency ("C1 to Center"). If you detect a point of interest in the spectrum diagram, you can place the cursor on it and then move the center frequency to the position of the cursor automatically ("Center to C1").

See Chapter 8.1, "Cursor measurements", on page 288.

Magnitude vs. phase display

The result of an FFT analysis is a spectrum of frequencies. Either the magnitudes or the phases of those frequencies are displayed, depending on the used FFT function. In "Basic" mode, and for the "Advanced" mode FFT functions |FFT|, FFT (re) and FFT (im), the magnitude is displayed. For the "Advanced" mode FFT (φ) function, the phase is displayed.

For magnitude display, you can select the scale and range of magnitudes to be displayed. For linear scaling, the vertical value range of the input signal is used. For logarithmic scaling, the logarithmic power of the frequency is displayed. In this case, the input signal must be given in either Volt or Watt. The resulting value range is defined by a maximum value and a range size. Logarithmic scaling can also be set in relation to a given reference value.

For phase display, you can select the unit and suppress phases beneath a threshold value which are most likely caused by noise. The value range [-π, + π] or [-180°, +180°] is used. Phase shifts due to a limitation of the value range can be eliminated using the "Unwrap" function.

Dependencies between FFT parameters

FFT analysis in the R&S RTO is highly configurable. Several parameters, including the resolution bandwidth, frequency span and center frequency, can be defined according to your requirements. Note, however, that several parameters are correlated and not all can be configured independently of the others.

The **resolution bandwidth** defines the minimum frequency separation at which the individual components of a spectrum can be distinguished. Small values result in high precision, as the distance between two distinguishable frequencies is small. Higher values decrease the precision, but increase measurement speed.

The minimum achievable RBW depends on the integration time which is equivalent to the number of samples available for FFT calculation. If a higher spectral resolution is required, the number of samples must be increased by using a higher sample rate or longer record length. To simplify operation some parameters are coupled and automatically calculated, such as record length and RBW.

The **frequency span** and **center frequency** define the start and stop frequency of the spectral diagram. By default, a suitable frequency range according to the resolution bandwidth is selected, in respect to performance and precision. Span and RBW settings are coupled, so that the parameters can be adjusted automatically as necessary.

With a **Span/RBW ratio** of 100 and a screen resolution of 1000 pixels, each frequency in the spectrum is displayed by 10 pixels. A span/RBW ratio of 1000 provides the highest resolution. For full flexibility, the span/RBW coupling can also be disabled. Note, however, that a higher span/RBW ratio (i.e. low RBW values and large frequency spans) result in large amounts of data and extend the duration of the calculation.

Advanced FFT functions

In "Advanced" math definition mode, other FFT results than the basic frequency magnitude can be displayed.

- **FFT (φ)**: phase display
- **FFT (im)**: imaginary part of FFT value (magnitude)
- **FFT (re)**: real part of FFT value (magnitude)

- **FFT -dφ*df** (group delay): the negative derivative of the phase with respect to frequency; useful to measure phase distortion

9.1.2 Configuring spectrum waveforms

During FFT analysis, a signal in the time domain is converted to a spectrum of frequencies. A basic spectrum waveform can be displayed quickly. By defining additional FFT parameters, the waveform can be configured in more detail.

As a result, either the magnitude or the phase of the determined frequencies can be displayed, or more complex FFT functions. Analysis can be restricted to an extract of the original time base, and the results display can be restricted to a specified frequency range.

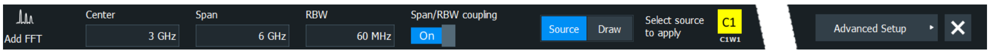
To display a basic spectrum waveform using the toolbar

1. Add the "FFT" icon to the toolbar, see Chapter 3.3.6.2, "Configuring the toolbar", on page 57 .

2. Tap the "FFT" icon on the toolbar.

   The FFT toolbar assist opens.

3. Tap the waveform for which the FFT is performed.

   The first available math waveform is configured to use the selected waveform as a source and the "Mag(FFT(x))" operator, and is enabled. The spectrum waveform is displayed in a new diagram.

To display a basic spectrum waveform using the [Math] key

1. Press the [FFT] key to open the "FFT" tab of the "Math" dialog box.
2. Set the "Source" to the input signal.
3. Enable "Display".
4. If necessary, edit the spectrum waveform parameters as described in the following procedures.

To display advanced spectrum waveforms

Other FFT results than the basic frequency magnitude can be displayed with the "Formula Editor".

1. Open the "Menu" > "Math" dialog.
2. Select the Equation tab.
3. Double-tap the editing area.

The "Formula Editor" is displayed.

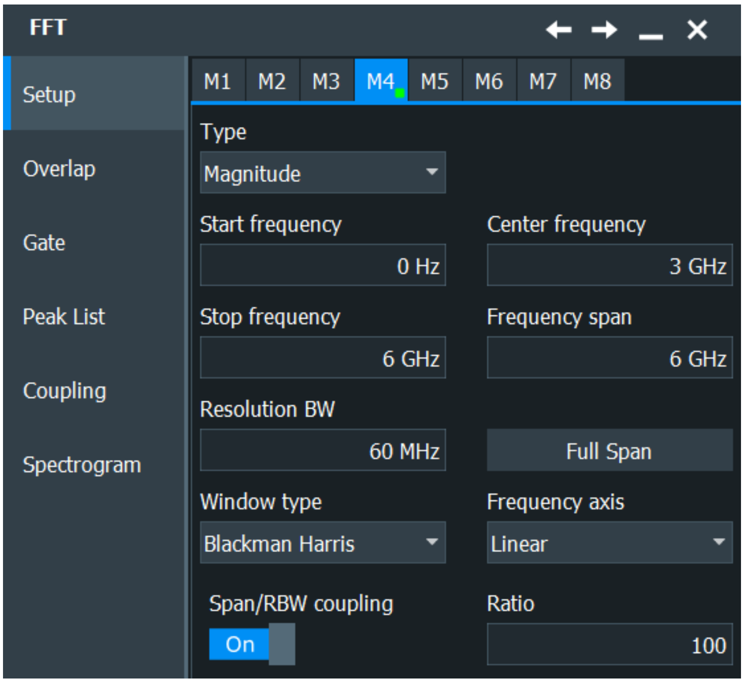
4. Delete the contents of the edit field.
5. Tap the "More" key to display further functions in the editor.
6. Tap the required FFT function key.
7. Select the source channel.
8. Close the parenthesis.
9. Tap "Enter"

To configure the spectrum of FFT analysis

By default, a suitable frequency range for the expected horizontal values according to the resolution bandwidth is selected, in respect to performance and precision. Span and RBW settings are coupled. If a more precise evaluation is required, for example for postprocessing in a different application, disable the coupling and change the frequency ranges and resolution bandwidth values as required.

1. Open [FFT] > "FFT setup".

   The "FFT" dialog opens.



2. In the "Setup" tab, tap the "Frequency axis" button to select the type of scaling you want to use: linear or logarithmic.
3. Disable the "Span/RBW coupling".
4. Specify the frequency range you want to display using one of the following methods:
   - Enter a "Center frequency" and a "Frequency span" that define the spectrum.

- Enter a "Start frequency" and "Stop frequency" that define the spectrum.
- Tap the "Full Span" button to display the complete spectrum resulting from the FFT analysis.

5. Define the resolution bandwidth for the FFT result.
   The resolution bandwidth defines how precise the results are, i.e. how close together the individual frequencies can be. Small values result in high precision, as the distance between two distinguishable frequencies is small. Higher values decrease the precision, but increase performance.

   You can define the RBW manually, or couple it to other FFT settings. Do one of the following:
   - To couple the RBW to the span, enable the "Span/RBW coupling" option and define the "Ratio". The smaller the ratio, the higher the RBW becomes to display the same frequency span.
   - Enter the "Resolution BW" manually. The "Span/RBW coupling" option is automatically disabled.
   - To couple the RBW to the specified record length, in the " Gate" tab of the "Advanced FFT" dialog box, select the "Record length controlled" option. This option is only available if no gate is being used ("Use Gate" disabled).
6. Select the most suitable "Window type" for your source data. Window functions are multiplied with the input values and thus can improve the FFT display. For details, see "Window type" on page 382.
7. Optionally, select an arithmetic mode for the FFT segments. This mode defines how the individual segment results are combined to a final spectrum waveform.
   In the "Overlap" tab of the "Advanced FFT" dialog box, tap "Segment arithmetic" and select the required mode from the list.
8. If you use an arithmetic mode, increase the "Overlap factor" for neighboring segments to increase the accuracy of the results.

To restrict the input values (gating)

By default, the FFT is calculated for the entire record length as defined for the data acquisition. However, you can restrict the time range for which the FFT is calculated, resulting in a restricted spectrum. Alternatively, the record length can be determined automatically according to the selected RBW.

1. Open [FFT] > "FFT setup" > "Gate" tab.
2. Determine how the input length is configured by selecting one of the following options:
   - To ensure that the FFT is calculated for the full defined record length, select the "Record length controlled" option. This option is only available if no gate is being used ("Use Gate" disabled). The RBW is adapted so that the record length can be acquired in the specified acquisition time. However, the RBW is restricted, so that data acquisition may fail if the record length is too long for the specified acquisition time.

- To couple the used record length to the required RBW, select the "RBW controlled" option. This option is only available if no gate is being used ("Use Gate" disabled).
  The required acquisition time for the defined RBW value is indicated.
- To restrict the basis of the FFT calculation to a certain time base, configure a time gate, that is: an extract of the time base in the original diagram. To do so, enable the "Use Gate" option, then do one of the following:
  – Select the "Absolute" mode and enter the "Start" and "Stop" times that define the gate area.
  – Select the "Relative" mode and enter the percentages of the total time base that define the "Relative Start" and "Relative Stop" times.
  – If a zoom area has already been defined in the original diagram and you want to use the same time base for FFT analysis, select "Zoom coupling".

The spectrum waveform displays the spectrum for the specified time span.

To configure magnitude results

1. Open [FFT] > "FFT setup" > "Setup" tab.
2. Set "Type" = "Magnitude".
3. In the "Scale" tab of the "Math" dialog, select the "Unit". Use logarithmic scaling only for input values in Volt or Watt.
4. Select the "Vertical scale mode" to configure the value range manually or use the automatic settings.
5. For "Vertical scale mode" > "Manual", define the size of the "Vertical range" and the "Vertical maximum" to be displayed.
   For "Vertical scale mode" > "Auto", define the size of the "Vertical range" to be displayed.
   For logarithmic scaling ("Unit = dB"), also define the "Reference level" to be used.

To configure phase results

1. Open [FFT] > "FFT setup" > "Setup" tab.
2. Set "Type" = "Phase".
3. In the "Scale" tab of the "Math" dialog, select the "Unit".
4. To eliminate phase shifts due to a limitation of the value range, enable the "Unwrap" function.
5. To suppress small phase values due to noise, enable the "Suppression" function and enter a "Threshold" value.

To couple spectrum displays

The settings for one or more spectrums can be coupled. Thus, if any FFT setting for any of the coupled spectrums are changed, they are changed for all coupled spectrums.

1. Open [FFT] > "FFT setup" > "Overlap" tab.
2. Select the spectrums that you need to couple. You cannot select the spectrum for the currently selected math waveform. Its settings are applied to the selected spectrums.
3. If necessary, define an FFT function to be used for the coupled math waveforms so that a spectrum is displayed. See Chapter 7.3.1, "Displaying math waveforms", on page 259.

9.1.3 FFT configuration settings

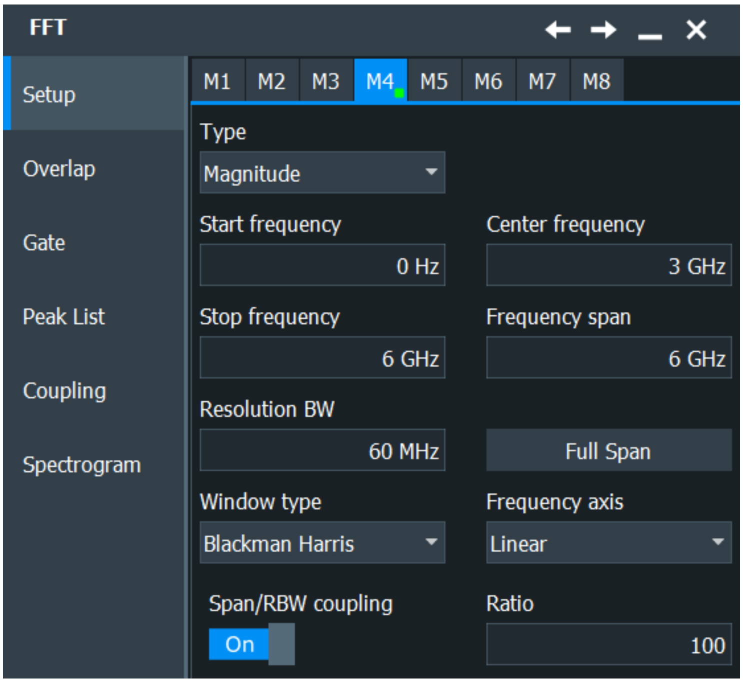
9.1.3.1 FFT setup

Access: [FFT] > "FFT setup" > "Setup" tab

In this tab, you define the settings for the FFT window. The display can be restricted to the results for a certain time base extract and to a specified frequency range.

Additional settings are available on this tab if the Spectrum Analysis option (R&S RTO-K18) is installed.

See Chapter 9.2.4, "Spectrogram configuration settings", on page 390.



Type

Selects between the magnitude and phase spectrum type.

Remote command:

`CALCulate:MATH<m>:FFT:TYPE` on page 1566

Display

If activated, a diagram for the defined math waveform is displayed on the touch screen.

Remote command:

`CALCulate:MATH<m>:STATe` on page 1458

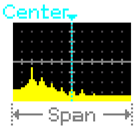
Frequency axis (R&S RTO-K18 only)

Defines the scaling method for the frequency (x-) axis of the spectrogram.

"Logarithmic"	Logarithmic scaling
"Linear Unit"	Linear scaling

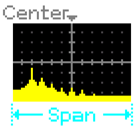
Remote command:
CALCulate:MATH<m>:FFT:LOGScale on page 1564

Center frequency
Defines the position of the displayed frequency range, which is (Center - Span/2) to (Center + Span/2). The width of the range is defined using the "Frequency span" setting.



Remote command:
CALCulate:MATH<m>:FFT:CFRequency on page 1565

Frequency span
The span is specified in Hertz and defines the width of the displayed frequency range, which is (Center - Span/2) to (Center + Span/2). The position of the span is defined using the "Center frequency" setting.



Remote command:
CALCulate:MATH<m>:FFT:SPAN on page 1566

Full span
Displays the full frequency span.

Remote command:
CALCulate:MATH<m>:FFT:FULLspan on page 1565

Start frequency
Defines the start frequency of the displayed frequency span.

Remote command:
CALCulate:MATH<m>:FFT:STARt on page 1564

Stop frequency
Defines the stop frequency of the displayed frequency span.

Remote command:
CALCulate:MATH<m>:FFT:STOP on page 1565

Span/RBW Coupling
Couples the frequency span to the "Resolution BW" setting.

Remote command:
CALCulate:MATH<m>:FFT:BANDwidth[:RESolution]:AUTO on page 1567

Span/RBW Ratio

Defines the coupling ratio for Span/RBW. This setting is only available if `CALCulate:MATH<m>:FFT:BANDwidth[:RESolution]:AUTO` is ON.

Remote command:
`CALCulate:MATH<m>:FFT:BANDwidth[:RESolution]:RATio` on page 1567

Resolution BW

Defines the resolution bandwidth. Note that the resolution bandwidth is correlated with the span, record length and acquisition time. If a constant record length is to be used, the RBW may be adapted if the required number of samples cannot be acquired. If span and RBW values are coupled, changing the span will also change the RBW.

For details see Chapter 9.1.1, "Fundamentals of FFT analysis", on page 371.

Remote command:
`CALCulate:MATH<m>:FFT:BANDwidth[:RESolution][:VALue]` on page 1567
`CALCulate:MATH<m>:FFT:BANDwidth[:RESolution]:ADJusted?` on page 1566

Window type

Windowing helps minimize the discontinuities at the end of the measured signal interval and thus reduces the effect of spectral leakage, increasing the frequency resolution.

Various different window functions are provided in the R&S RTO to suit different input signals. Each of the window functions has specific characteristics, including some advantages and some trade-offs. Consider these characteristics carefully, to find the optimum solution for the measurement task.

Window type	Frequency resolution	Magnitude resolution	Measurement recommendation
Rectangular	Best	Worst	Separation of two tones with almost equal amplitudes and a small frequency distance
Hamming Hann	Good	Poor	Frequency response measurements, sine waves, periodic signals and narrow-band noise
Blackman Harris (default)	Worst	Best	Mainly for signals with single frequencies to detect harmonics Accurate single-tone measurements
Gaussian	Good	Good	Weak signals and short duration
Flattop2	Poor	Best	Accurate single-tone measurements
Kaiser Bessel	Poor	Good	Separation of two tones with differing amplitudes and a small frequency distance

Remote command:
`CALCulate:MATH<m>:FFT:WINDow:TYPE` on page 1567

Use color table

If enabled, the spectrum waveform (and a spectrogram, if available) is displayed according to the assigned color table. For information on the available color tables, see Chapter 4.3.2, "Color tables", on page 81.

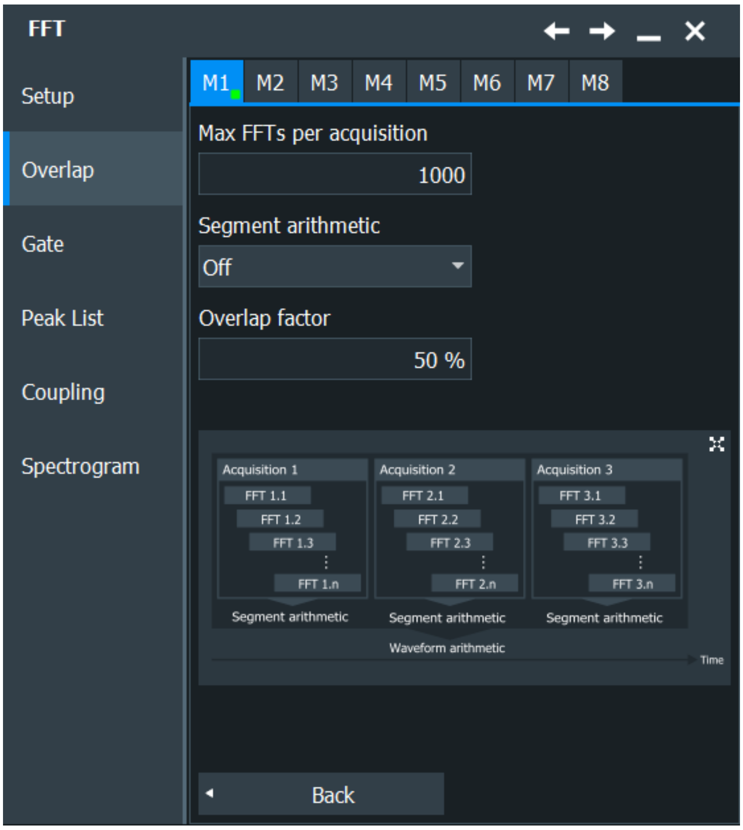
If this option is disabled, the preset color of the selected channel source is displayed, and the intensity of the specific signal color varies according to the cumulative occurrence of the values. For spectrum diagrams, this setting corresponds to the common waveform display. The spectrogram, on the other hand, is then displayed in gray tones, which is not useful.

Remote command:
`CALCulate:MATH<m>:FFT:USEColtab` on page 1578

9.1.3.2 FFT overlap

Access: [FFT] > "FFT setup" > "Overlap" tab

In this tab, you define the settings for the magnitude and phase of the frequencies.



Max FFTs per acquisition
Restricts the maximum number of FFTs to be calculated for each data acquisition. Due to the other parameter settings, the required number of FFTs may become very high, thus slowing performance. By restricting the number of FFTs, you can avoid performance loss without changing the other parameters.

However, if the maximum number of FFTs is lower than the required number to cover the entire waveform, the waveform may only be analyzed partially. In this case, the "Frame coverage" indicates the percentage of the waveform that was analyzed, i.e. which part of the data was included in the FFT calculation.

Remote command:
`CALCulate:MATH<m>:FFT:FRAMe:MAXCount` on page 1569
`CALCulate:MATH<m>:FFT:FRAMe:COVerage?` on page 1569

Segment arithmetic
FFT analysis can only be performed on a maximum number of values at once. If more values must be calculated, the input signal is divided into segments, each of which is calculated separately. The segments need not be disjunct, that is: they may overlap, so that some values have several FFT results. In this case, the arithmetic mode defines how the final result is calculated from the individual results.

The following methods are available:

"Off"	The data of only one segment is considered. In effect, no arithmetics are processed.
"Envelope"	Detects the minimum and maximum values for FFT calculation over all segments. The resulting diagram shows two envelope waveforms: the minimums (floor) and maximums (roof). These envelopes indicate the range of all FFT values that occurred.
"Average"	The average is calculated over all segments.
"RMS"	The root mean square is calculated over all segments. The result is the average power spectrum. If you measure the channel power on this RMS spectrum, you get the same result as for the average channel power measurement on segments.
"MinHold"	Determines the minimum result for each input value from the data of the current acquisition and the acquisitions before. Only available if option R&S RTO-K18 is installed.
"MaxHold"	Determines the maximum result for each input value from the data of the current acquisition and the acquisitions before. Only available if option R&S RTO-K18 is installed.

Remote command:
`CALCulate:MATH<m>:FFT:FRAMe:ARIThmetics` on page 1569

Overlap Factor
Defines the minimum factor by which two neighboring segments overlap. If the required number of segments to cover the input values allows for more overlap, the factor is increased.

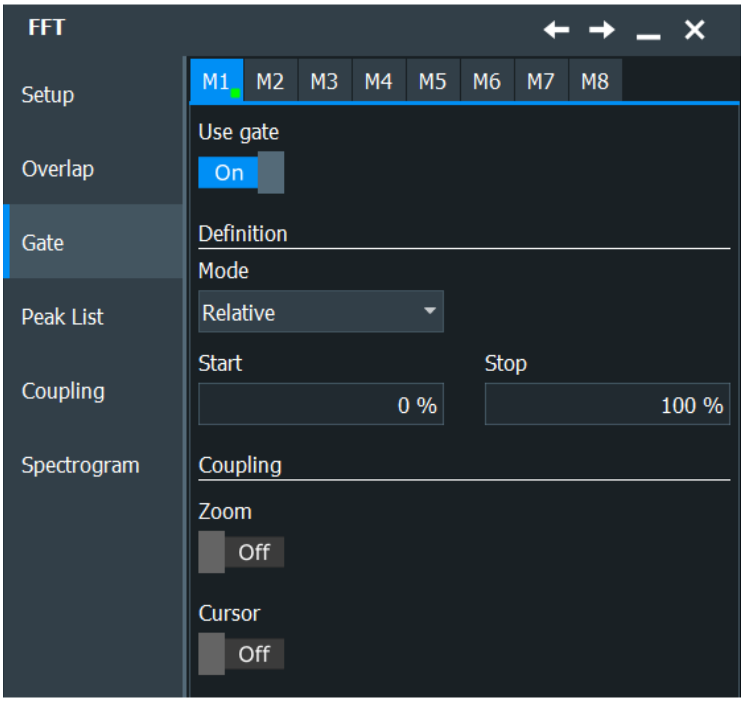
The higher the overlap factor, the more segments are used. This leads to more individual results and improves detection of transient signal effects. However, it also extends the duration of the calculation.

Remote command:
`CALCulate:MATH<m>:FFT:FRAMe:OFACtor` on page 1570

9.1.3.3 FFT gating

Access: [FFT] > "FFT setup" > "Gate" tab

FFT gating allows you to restrict FFT analysis to a certain time base of the input signal.



If no gate is used, you can define the record length as dependent on the RBW, or the RBW as dependent on the record length (which is defined by the acquisition time).

Use Gate

Enables FFT gating and shows the gate.

If enabled, the "Gate Definition" settings are used.

If disabled, the relation between the record length and the RBW can be defined manually in the "Setup" tab instead.

When a gate is used, the RBW is adapted, if necessary. The smaller the gate, the higher the RBW.

For details, see Chapter 9.1.1, "Fundamentals of FFT analysis", on page 371.

Gate Definition

Defines the gate settings for FFT gating.

Zoom ← Gate Definition
Zoom coupling is available if a zoom is defined. As long as "Zoom coupling" is enabled, the gate area is defined identically to the zoom area - if you change the zoom, the gate changes as well.

If several zoom diagrams are defined, select the zoom diagram to be used for gating. The "Start" and "Stop" values of the gate are adjusted accordingly.

Zoom coupling can be set for measurement gates, FFT gates, and search gates. The zoom must be defined on the diagram that contains the signal source of the measurement, FFT, or search.

Remote command:
`MEASurement<m>:GATE:ZCOupling` on page 1546
`MEASurement<m>:GATE:ZDIagram` on page 1547
`CALCulate:MATH<m>:FFT:GATE:ZCOupling` on page 1572
`SEARch:GATE:ZCOupling` on page 1631
`SEARch:GATE:ZDIagram` on page 1631

Gate description ← Gate Definition
Defines whether the gate settings are configured using absolute or relative values.

"Absolute" The gate is defined by absolute start and stop values.

"Relative" The gate's start and stop values are defined by a percentage of the value range.

Remote command:
`CALCulate:MATH<m>:FFT:GATE:MODE` on page 1571
`MEASurement<m>:GATE:MODE` on page 1545
`SEARch:GATE:MODE` on page 1629

(Relative) Start ← Gate Definition
Defines the starting value for the gate.

Remote command:
`CALCulate:MATH<m>:FFT:GATE:ABSolute:STARt` on page 1571
`CALCulate:MATH<m>:FFT:GATE:RELative:STARt` on page 1572
`MEASurement<m>:GATE:ABSolute:STARt` on page 1545
`MEASurement<m>:GATE:RELative:STARt` on page 1545
`SEARch:GATE:ABSolute:STARt` on page 1630
`SEARch:GATE:RELative:STARt` on page 1630

(Relative) Stop ← Gate Definition
Defines the end value for the gate.

Remote command:
`CALCulate:MATH<m>:FFT:GATE:ABSolute:STOP` on page 1571
`CALCulate:MATH<m>:FFT:GATE:RELative:STOP` on page 1572
`MEASurement<m>:GATE:ABSolute:STOP` on page 1545
`MEASurement<m>:GATE:RELative:STOP` on page 1545
`SEARch:GATE:ABSolute:STOP` on page 1630
`SEARch:GATE:RELative:STOP` on page 1631

Record Length/RBW Coupling
The record length and resolution bandwidth are coupled during FFT analysis. If you change one value, the other must be adapted accordingly. You can keep either value constant, thus preventing automatic adaptation when the other parameter is changed. However, this may cause the FFT analysis to fail.

This setting is only available if gating is not enabled (otherwise the gate determines the RBW automatically).

For details, see Chapter 9.1.1, "Fundamentals of FFT analysis", on page 371.

"Record length controlled"	The record length remains constant. If not enough samples are available for the selected RBW, the RBW is decreased.
"RBW controlled"	The RBW is not adapted, i.e. remains as defined. The required acquisition time for this RBW is indicated. If necessary and possible, the record length is extended to acquire the required number of samples.

Remote command:
`CALCulate:MATH<m>:FFT:GATE:COUPling` on page 1570

Required acquisition time
The required acquisition time is calculated for the defined RBW value if "RBW constant" is selected, and is displayed for information only. If the required acquisition time is not available (e.g. because acquisition has already been stopped), an error message is displayed in the FFT setup tab indicating that not enough samples are available for the defined RBW.

Remote command:
`TIMebase:RACTime?` on page 1571

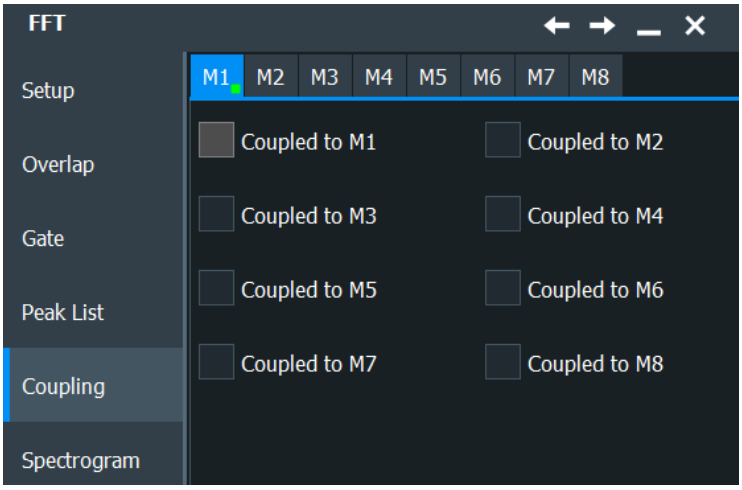
9.1.3.4 Peak list

For a description of the "Peak List" settings, see Chapter 8.2.7, "Spectrum measurements", on page 338.

9.1.3.5 FFT coupling

Access: [FFT] > "FFT setup" > "Coupling" tab

Up to four spectrum displays can be shown simultaneously, one for each math waveform. The settings for one or more spectrums can be coupled. Thus, if any FFT setting for any of the coupled spectrums are changed, they are changed for all coupled spectrums.



Coupled to M1...M8

Copies the current FFT settings of the selected math waveform (M1...M8) to the other selected math waveforms, and couples those waveforms. Thus, if any FFT setting for any of the coupled spectrums are changed, they are changed for all coupled spectrums.

Two different sets of spectrums can be coupled at the same time, for instance "M1" can be coupled to "M2", while "M3" is coupled to "M4".

Note that the formula of the coupled math waveforms is not changed. If necessary, you must select an FFT function for the math waveform manually before the FFT settings of the coupled waveform are applied. See Chapter 7.3.1, "Displaying math waveforms", on page 259.

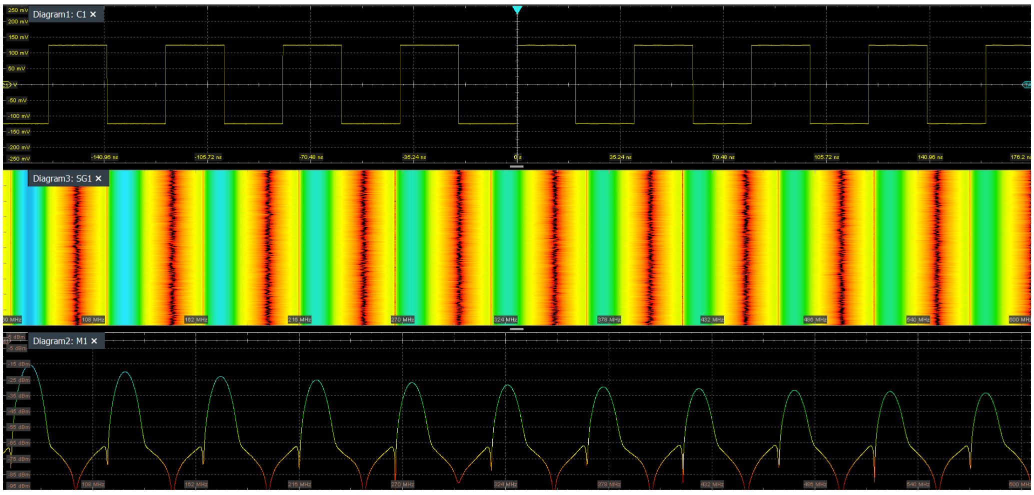
Remote command:
`CALCulate:MATH<m>:FFT:COUPled:WITH<1..8>` on page 1576

9.2 Spectrum Analysis (option R&S RTO-K18)

This chapter describes the Spectrum Analysis option R&S RTO-K18.

9.2.1 Spectrogram display

The Spectrum Analysis option provides a new diagram for spectrum waveforms: a spectrogram. When you enable a spectrogram, three windows are displayed: the power vs. time diagram at the top, the spectrogram in the middle (labeled "SG") and the power vs. frequency (=spectrum) diagram at the bottom.



A spectrogram shows how the spectral density of a signal varies over time. The x-axis shows the frequency, the y-axis shows the time. A third dimension, the power level, is indicated by different colors. Thus you can see how the strength of the signal varies over time for different frequencies.

The spectrogram is updated with each data acquisition, from top to bottom, so that the most recent trace is at the bottom. Up to two time lines can be shown at a specified position so that you can analyze the spectrum at a specific point in time.

The spectrum diagram indicates the power vs. frequency values for a single data acquisition. If a time line is enabled, the spectrum shows the results at the selected time. Otherwise, the spectrum shows the results of the most recent data acquisition.

9.2.2 Spectrum Analysis functions

In addition to spectrograms, the Spectrum Analysis option also provides some new automatic measurements based on spectrum waveforms.

- A peak list measurement detects all peaks above a user-definable threshold and optionally indicates the peaks in the spectrum diagram.

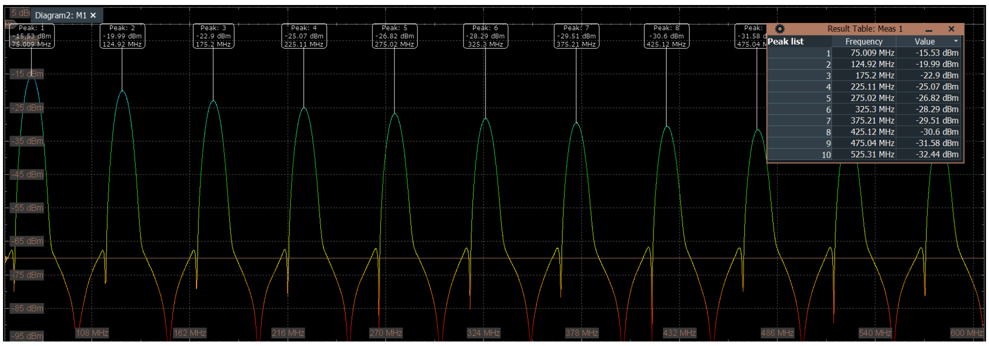


Figure 9-1: Peak list with labels for spectrum waveform

For a description of the measurement settings, see Chapter 8.2.7.2, "Settings for spectrum measurements", on page 339.

- The THD measurements are an extension to the basic THD measurement. See Chapter 8.2.7, "Spectrum measurements", on page 338 for details.
- Cursor measurements on spectrum waveforms provide easy center definition and peak search functions, see Chapter 8.1.1.2, "Cursor measurements on spectrum waveforms", on page 289.

9.2.3 Configuring spectrograms

Spectrograms are only available if the Spectrum Analysis option R&S RTO-K18 is activated.

1. Tap the "Spectrogram" icon on the toolbar.

   The "Spectrogram" toolbar assist opens.

2. Select the source of the spectrogram.

   A spectrogram diagram is displayed. A new signal icon for the spectrogram is displayed on the signal bar ("SGx").
   If the selected source is a channel waveform, an FFT is started, on which the spectrogram is created.
   Additional settings for time lines become available in the dialog box.
3. Optionally, to display a time line and thus mark a specific waveform in the spectrogram, select "Enable" for one of the two time lines.

   A small arrow icon labeled "T1" / "T2" indicates the position of the time line in the spectrogram.
   The spectrum diagram displays the results for the selected waveforms. A new signal icon is displayed on the signal bar for each time line ("SGxTL1|2").
4. To view the spectrum for each time line in a separate diagram, drag the signal icon for one time line to the diagram area and drop it.

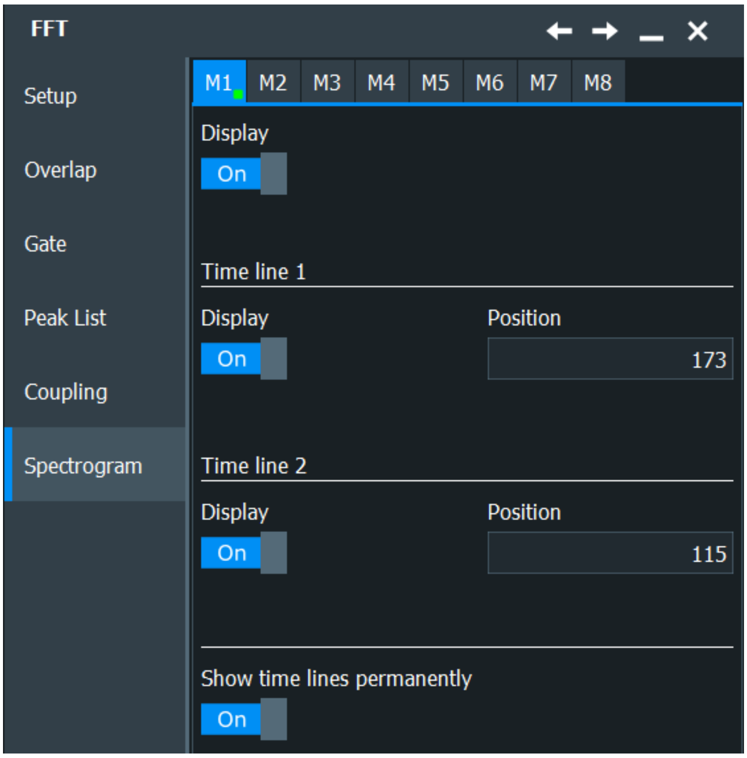
   A new window is displayed for the selected time line, and the original diagram displays the other time line.
5. To view a different waveform from the spectrogram, move the time lines in the spectrogram.

9.2.4 Spectrogram configuration settings

Access: [FFT] > "FFT setup" > "Spectogram" tab

Spectrograms are only available if the option R&S RTO-K18 is activated. Furthermore, a math (FFT) waveform must be configured and enabled.

See Chapter 9.1.2, "Configuring spectrum waveforms", on page 375.



Display

Enables the spectrogram display.

If enabled, a new signal icon for the spectrogram is displayed on the signal bar ("SGx").

Remote command:
`CALCulate:MATH<m>:FFT:SPECtrogram:STATe` on page 1578

Time line 1/2

A time line marks a single spectrum in the spectrogram, that is: the power vs frequency results for the data acquired at a specific time. After enabling a time line, the results for that time are displayed in the spectrum diagram. A small arrow icon labeled "T1" / "T2" indicates the position of the time line in the spectrogram. A new signal icon is displayed on the signal bar for each time line ("SGxTL1|2").

You can enable and display two time lines at the same time. This allows you to compare the results at different times.

The position of the time line is defined by the index of the data acquisition in the history. How many acquisitions are available depends on the history settings.

See "Player" on page 280.

Remote command:
`CALCulate:MATH<m>:FFT:SPECtrogram:TIMeline<n>:STATe` on page 1579
`CALCulate:MATH<m>:FFT:SPECtrogram:TIMeline<n>:POSition` on page 1579

Show lines permanently
Displays the spectrogram time lines in the diagrams until you disable this option.

If disabled, only the small arrow icons are permanently visible. The line is only displayed temporarily when you touch the arrow.

10 Mask testing

10.1 About mask testing

Masks are used to determine whether the signal remains within specified limits, e.g. to uncover signal anomalies or test compliance and stability of digital signals. The limits are specified as "mask", which is laid over the input signal in the display. Thus you can easily detect where the signal violates the mask.

Mask testing with R&S RTO has only a minor impact on the acquisition rate, thus mask violations are detected fast and reliably.

With R&S RTO, you can define own masks easily. Specific actions can be executed when mask violations occur. For error analysis, you can stop the acquisition on a failed test and use the history view to look at the previous waveforms.

Mask test

A mask test consists of:

- Mask definition
- Waveform to be tested
- Fail criteria for test
- Actions to be taken on violation or successful completion

Mask Definition

A mask can be created in several ways:

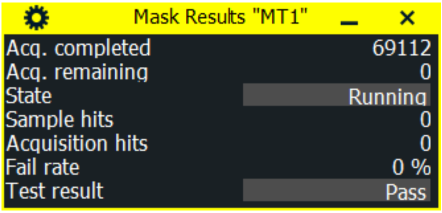
- The individual mask points are defined, either on the touch screen or as numerical values. This mask type is called *user mask*.
  For details, see Chapter 10.2.2.1, "Mask definition: user mask", on page 397
- The mask is derived from an existing waveform. This mask type is called *waveform mask*.
  For details, see Chapter 10.2.2.2, "Mask definition: waveform mask", on page 400.

Fail Criteria for Testing

The fail criteria for a mask test is set by two parameters: "Fail condition" and "Violation tolerance". Fail condition defines if sample hits or the number of acquisitions with sample hits are considered. Violation tolerance sets the number of tolerable sample hits or acquisition hits. A test has failed if the number of sample hits or acquisition hits exceeds the limit of violation tolerance hits.

10.1.1 Results of a mask test

The result box of a mask test shows the following test results:



Acq. completed

Number of tested acquisitions.

Remote command:
MTESt:RESult:COUNt:WAVeforms? on page 1595

Acq. remaining

Remaining acquisitions until "Average count / Nx Single count" is reached.

The value is useful if you test a specified number of acquisitions with action "Stop acquisition" on violation. Also if the acquisition has been stopped manually before the required number of acquisitions has been acquired.

Remote command:
MTESt:RESult:COUNt:REMaining? on page 1595

State

Shows if the test has been completed. The state is set to "Finished" when "Nx Single count" acquisitions are tested and the number of "Acq. remaining" is 0. as long as the number of tested acquisitions is less the "Nx Single count" number, the state is "Running".

If you run the acquisition with [Run Stop], or the number of played history acquisitions exceeds "Nx Single count", the mask testing is performed according to fail criteria settings independently of the test state. The testing is not stopped when the state is set to "Finished".

Remote command:
MTESt:RESult:STATe? on page 1595

Sample hits

Number of samples that hit the mask.

Remote command:
MTESt:RESult:COUNt:FAILures? on page 1596

Acquisition hits

Number of acquisitions that contained at least one sample hit.

Remote command:
MTESt:RESult:COUNt:FWAVeforms? on page 1596

Fail rate

Ratio of acquisition hits to the number of tested acquisitions.

Remote command:
MTESt:RESult:FRATe? on page 1596

Test result
A test has failed if the number of sample hits or acquisition hits exceeds the limit of "Violation tolerance" hits.

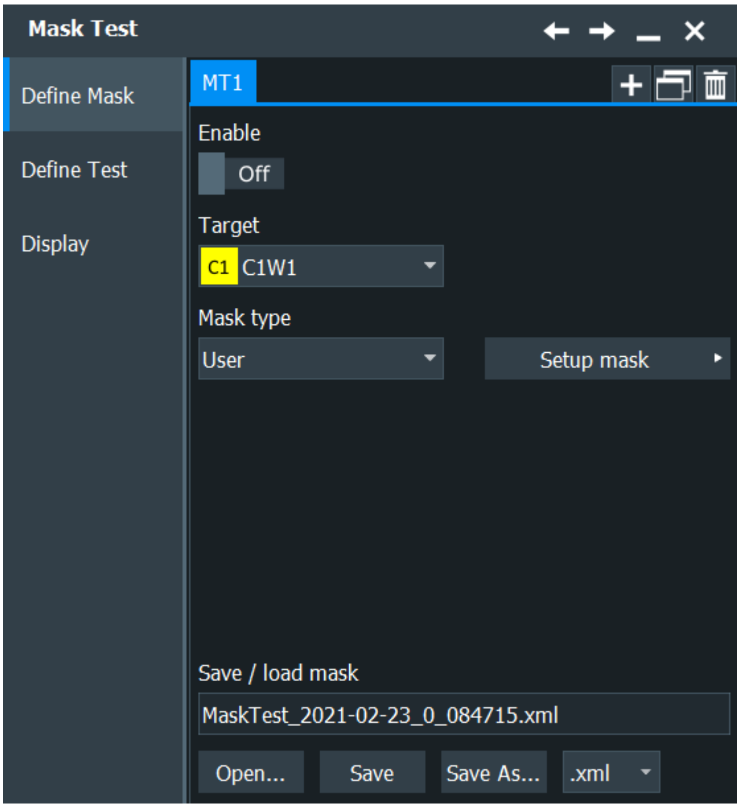
Remote command:
`MTESt:RESult[:RESult]?` on page 1595

10.2 Mask test settings

10.2.1 Define mask

Access: [App Cockpit] > "Analysis" tab > "Mask" > "Define Mask"

The "Define Mask" tab provides all settings for the mask test itself: the waveform to be tested,and saving/loading the mask definition.



The content of the "Test Definition" tab depends on the selected definition type. If "Waveform" is selected, the main mask settings can be set directly on the "Test Definition" tab. For a description of these settings, see Chapter 10.2.2.2, "Mask definition: waveform mask", on page 400.

Make sure that the correct "Mask Test" tab is selected on the left side before you enter the settings.

Remote commands:

`MTESt:ADD` on page 1580

`MTESt:REMove` on page 1581

Enable

Activates and deactivates the mask test. If the acquisition is running, the test starts immediately. Otherwise, the test starts when acquisition is started.

The testing is stopped when acquisition is stopped, or if a stop action is configured with "Stop acq." on page 405.

Closing the result box also disables the mask test.

Remote command:
`MTESt[:STATe]` on page 1581

Target

Selects the waveform to be tested against the mask. All channel waveforms can be tested.

Remote command:
`MTESt:SOURce` on page 1581

Mask Type

Sets the method of mask definition.

"User" The mask is created manually by tapping the mask points on the touch screen and/or by entering the numerical x- and y-values of the mask points.

"Waveform" The mask is created from an existing waveform. The waveform builds the upper and lower limit line of the mask, and the limits are moved and stretched. The result is a tolerance tube around the waveform that is used as mask.

"Eye" Requires jitter option R&S RTO-K12.
The mask is created by selecting the shape and setting its dimensions according to the test standard.

Remote command:
`MTESt:CTYPe` on page 1583

Setup mask

Opens a dialog for defining the mask. See Chapter 10.2.2, "Mask definition", on page 397.

Save / load mask

Provides all functions to store and recall a mask test. The mask definition, defined actions and fail conditions are stored in an R&S RTO-specific .xml file.

"Open"	Opens a dialog box where you can select the file name. See also: Chapter 12.5, "File selection dialog", on page 478.
"Save, Save As"	Stores the specified file.

Remote command:
MTESt:FILE:NAME on page 1583
MTESt:FILE:SAVE on page 1584
MTESt:FILE:OPEN on page 1584
MTESt:FILE:DELete on page 1584

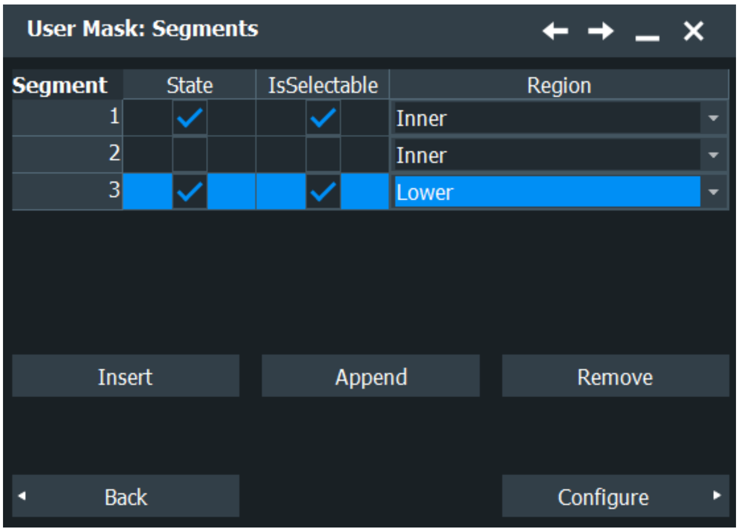
10.2.2 Mask definition

With mask definition, you define the shape of the mask - the form and position of its limit lines. The content of the "Mask Definition" tab depends on the selected Mask Type: "User" or "Waveform".

Below, you find the specific settings:

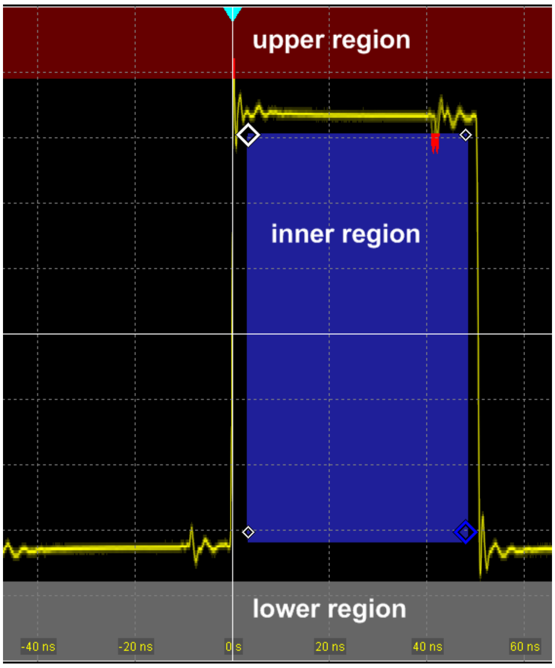
10.2.2.1 Mask definition: user mask

Access: [App Cockpit] > "Analysis" tab > "Define Mask" > "Mask Type" = "User" > "Setup mask"



A user mask is defined by entering the time and voltage values for all corner points of the mask segments. A user mask has at least one segment. Complex masks can have up to 16 segments.

An inner segment is an area defined by three or more points. Upper and lower segments limit the signal on top and bottom of the screen. They are defined by a line, the region above or below the line is set automatically as mask segment.



Alternatively, you can set the corner points on the touch screen and adjust the values in the "Mask Definition" tab.

To save the mask, select the "Define Mask" tab and save the mask test.

Make sure that the correct "Mask Test" tab is selected on the left side before you enter the settings.

Mask segments

Defines the number and state of mask segments for the selected mask test. Here you can:

- Insert a new segment before the selected segment.
- Append a new segment at the end of the list.
- Remove the selected mask segment from the list.
- Select the region that builds the mask.
  - Inner region: the segment points form a closed geometrical shape, which is the mask segment.
  - Upper region: the segment points are connected to a line, the display area above this line is the mask segment.
  - Lower region: the segment points are connected to a line, the display area below this line is the mask segment.
- Enable and disable the mask segments individually. Disabled segments are not considered by running tests.

Remote command:

`MTESt:SEGMent:STATe` on page 1585

`MTESt:SEGMent:ADD` on page 1585

`MTESt:SEGMent:REMove` on page 1586

`MTESt:SEGMent:INSert` on page 1585

MTESt:SEGMent:REGion on page 1586
MTESt:SEGMent:COUNt? on page 1585

Configure
Opens a dialog to define the selected segment.

Definition of segment
The number of the selected segment is shown above the table. In the definition table, the individual points of the selected mask segment are listed with exact horizontal and vertical numerical coordinates. Here you can:

- Insert a new point before the selected point.
- Append a new point at the end of the list.
- Remove the selected point from the list.
- Change the x- and y-values of each point. To scale or move the complete segment, use offset and factor values, see Rescale.

Remote command:
MTESt:SEGMent:POINt:ADD on page 1587
MTESt:SEGMent:POINt:REMove on page 1587
MTESt:SEGMent:POINt:INSert on page 1587
MTESt:SEGMent:POINt:X on page 1588
MTESt:SEGMent:POINt:Y on page 1588
MTESt:SEGMent:POINt:COUNt? on page 1587

Rescale
You can rescale and move mask segments by numerical input of factors and offsets.

The values change the selected mask segment and take effect on "Apply".

Offset X ← Rescale
Moves the mask segment horizontally. The specified offset is added to the x-values of all points of the selected mask segment.

To take effect, tap "Apply".

Remote command:
`MTESt:SEGMent:RESCale:XOFFset` on page 1589

Factor X ← Rescale
Stretches or compresses the selected mask segment in horizontal direction. The x-values of all points of the selected mask segment are multiplied with this factor. Factors >1 stretch the mask segment, while factors between 0 and 1 compress it. Negative values are possible and change the algebraic sign.

To take effect, tap "Apply".

Remote command:
`MTESt:SEGMent:RESCale:XFACtor` on page 1589

Offset Y ← Rescale
Moves the mask segment vertically. The specified offset is added to the y-values of all points of the selected mask segment.

To take effect, tap "Apply".

Remote command:
`MTESt:SEGMent:RESCale:YOFFset` on page 1590

Factor Y ← Rescale
Stretches or compresses the selected mask segment in vertical direction. The y-values of all points of the selected mask segment are multiplied with this factor. Factors >1 stretch the mask segment, while factors between 0 and 1 compress it. Negative values are possible and change the algebraic sign.

To take effect, tap "Apply".

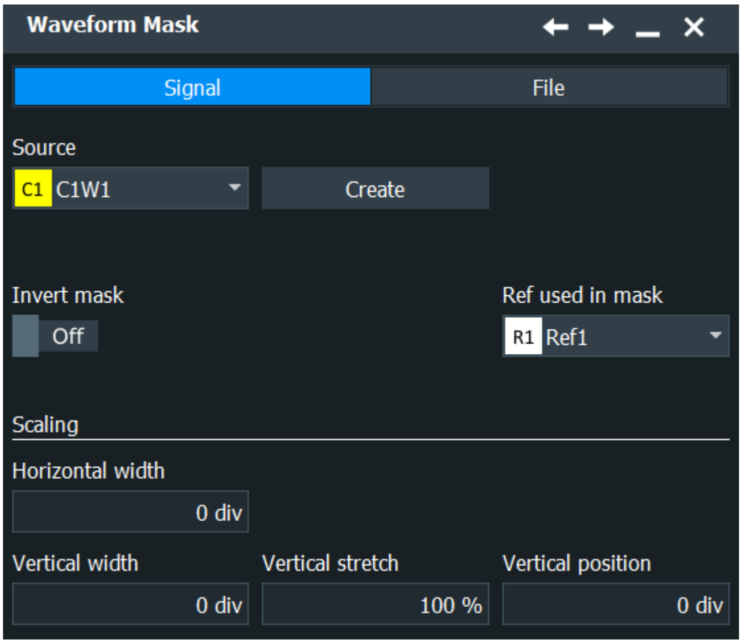
Remote command:
`MTESt:SEGMent:RESCale:YFACtor` on page 1589

Apply ← Rescale
Multiplies and adds the given x- and y-factors and offsets to the coordinates of all points of the selected mask segment.

Remote command:
`MTESt:SEGMent:RESCale:RECalculate` on page 1588

10.2.2.2 Mask definition: waveform mask

Access: [App Cockpit] > "Analysis" tab > "Define Mask" > "Mask Type" = "Waveform" > "Setup mask"



A waveform mask is created from an existing waveform. The waveform builds the upper and lower limit line of the mask, and the limits are moved and stretched. The result is a tolerance tube around the waveform that is used as mask.

During mask testing using a waveform mask, the record length is limited to 1 MSample.

The source for a waveform mask is a reference waveform. The reference waveform can be defined before mask definition, or loaded from a file, or it is created from the waveform to be tested.

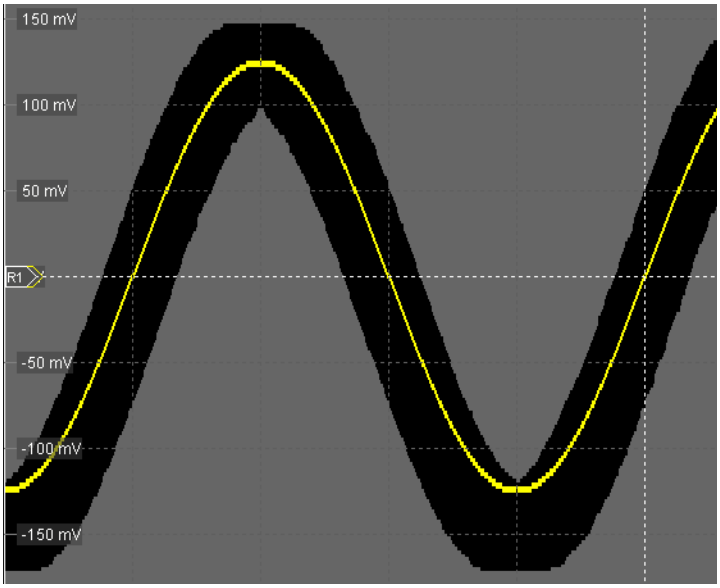


Figure 10-1: Waveform mask

Signal
The reference waveform is created from the selected "Source".

Sets the reference waveform from which the mask is created.

Create ← Signal
Creates the upper and lower mask limit from the "Source" and "Ref used in mark". If the reference waveform was not defined before, it is created automatically from the mask test "Source" waveform.

File
Loads the waveform from the selected file to the "Reference" and creates the mask immediately.

Invert mask
If enabled, the area inside the mask is highlighted, which the signal must not exit. If disabled, the area outside the mask limits is highlighted.

Horizontal width
Sets the width of the mask in horizontal direction. The specified number of divisions is added to the positive x-values and subtracted from the negative x-values of the mask limits in relation to the source waveform of the mask. The overall mask width is twice the specified horizontal width.

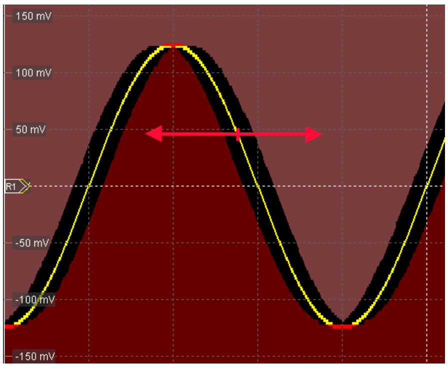


Figure 10-2: Waveform mask with horizontal width = 0.2 div

Remote command:
`MTESt:WFMRescale:XWIDth` on page 1591

Vertical width
Sets the width of the waveform mask in vertical direction. The specified number of divisions is added to the y-values of the upper mask limit and subtracted from the y-values of the lower mask limit. Thus, the upper half of the mask is pulled upwards, the lower half is pulled down, and the overall height of the mask is twice the vertical width.

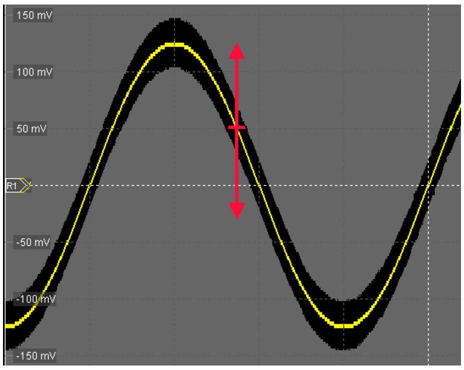


Figure 10-3: Waveform mask with vertical width = 0.5 div

Remote command:
MTESt:WFMRescale:YWIDth on page 1591

Vertical stretch
Sets the vertical scaling to stretch the mask in y-direction. The scaling axis is the horizontal line through the lowest value of the lower mask limit. Values > 100% stretch the mask, and values < 100% compress it.

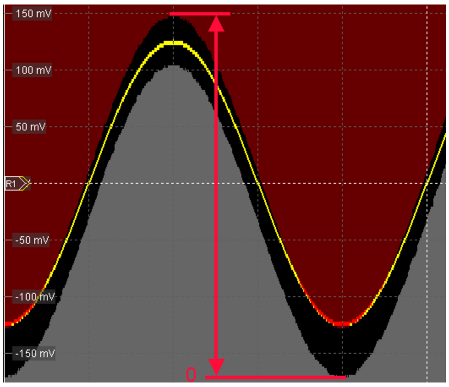


Figure 10-4: Waveform mask with vertical width = 0.5 div, vertical position = -0.5 div, vertical stretch = 110%

Remote command:
MTESt:WFMRescale:YSTRetch on page 1592

Vertical position
Moves the mask vertically within the display.

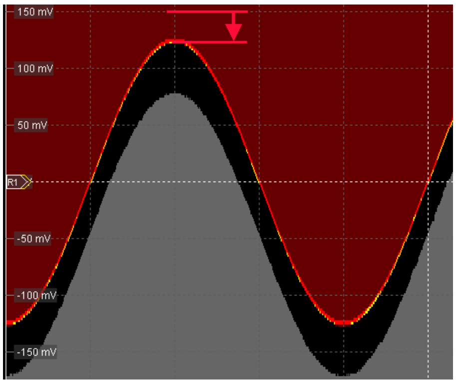


Figure 10-5: Waveform mask with vertical width = 0.5 div and vertical position = -0.5 div

Remote command:
`MTESt:WFMRescale:YPOSition` on page 1592

10.2.3 Actions on event

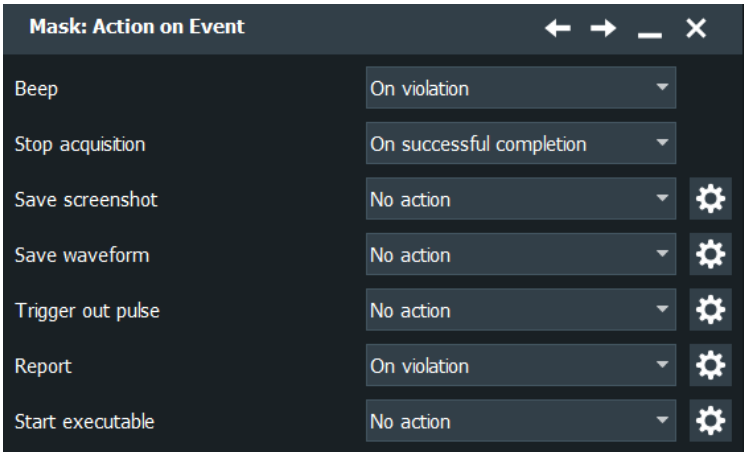
The settings in this tab define what happens when the mask test has failed or when it has passed successfully. Furthermore, you can reset all totals and results in the "Mask Test" result boxes.

Most actions can be initiated either on failure or on success:

- On violation
  The action is initiated when the fail criteria is fulfilled.
- On successful completion
  The action is initiated when the [Single] acquisition has finished and the fail criteria is not fulfilled - the fail condition and violation tolerance limit have not been reached.

There are two usual test practices:

- Testing a defined number of waveforms against the mask and initiate an action when the acquisition cycle has been completed without failure:
  - Set the number of acquisitions to be tested: "Average count (N-single count)"
  - Start [Single]
- Testing a continuous acquisition or a defined number of waveforms against the mask and initiate an action when the fail criteria is fulfilled



Make sure that the correct "Mask Test" tab is selected on the left side before you enter the settings.

Beep
Generates a beep sound.

Remote command:
`MTESt:ONViolation:BEEP` on page 1592

Stop acq.
Stops the waveform acquisition on mask violation.

Remote command:
`MTESt:ONViolation:STOP` on page 1593

Saves screenshot
Saves a screenshot including the mask test results.

Report
Creates and saves a report using the settings.

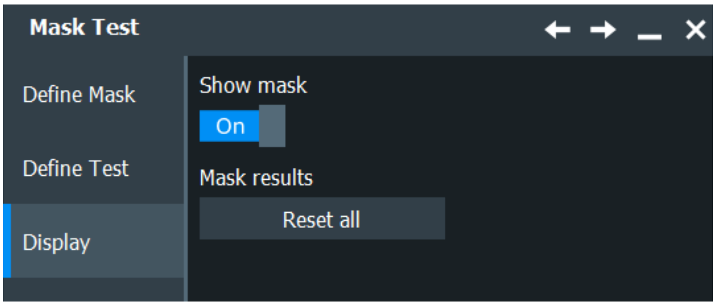
Remote command:
`MTESt:ONViolation:REPort` on page 1593

Start Executable
Starts an external application. Tap "Config Executable" to set the application path and parameters.

Remote command:
`MTESt:ONViolation:RUNexec` on page 1594

10.2.4 Mask display

Access: [App Cockpit] > "Analysis" tab > "Mask" > "Display".



In this dialog, you can reset the mask results and display the mask.

Show mask
Switches the display of all mask segments on or off.

Mask results > Reset all
Clears all totals and results in all "Mask Test" result boxes.

Remote command:
`MTESt:RST` on page 1581

10.2.5 Mask appearance

You can define additional mask appearance settings in the "Settings" > "Appearance" > "Mask" dialog.

See Chapter 4.3.10, "Mask appearance settings", on page 92

10.3 Mask testing on eye diagrams

To perform mask testing on eye diagrams, the option R&S RTO-K12 adds a special mask definition type to the common mask definitions: the definition type "Eye".

Using the "Eye" mask type, you can easily define eye masks of various shapes. You can also use "User" masks to define free mask shapes.

10.3.1 Test definition for eye mask tests

Access: [App Cockpit] > "Analysis" tab > "Define Mask" tab

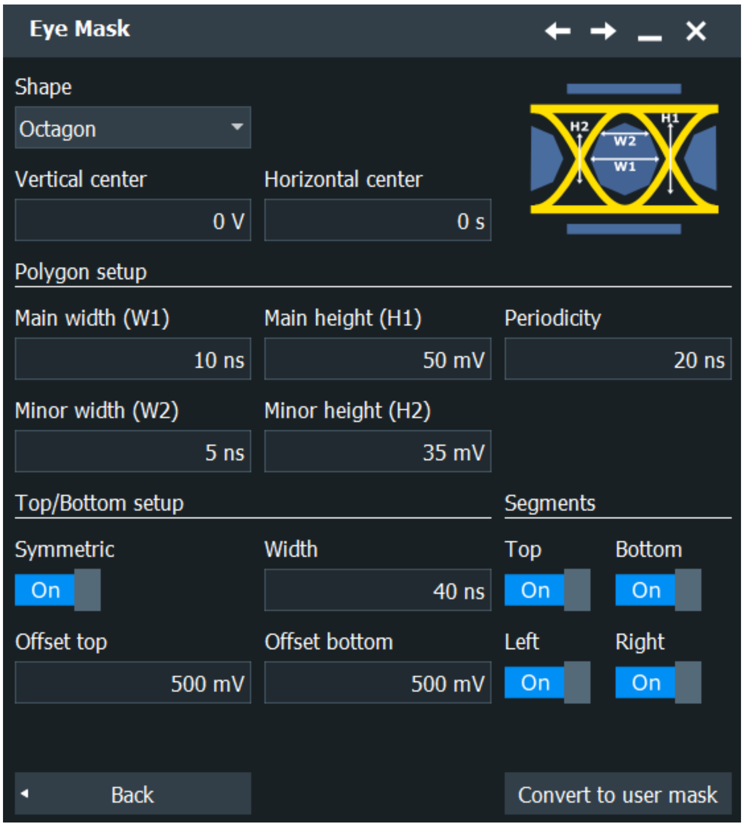
1. Select the "Source": the channel waveform that is analyzed for jitter.
2. Select the "Mask Type" = "Eye".
3. In the "Define Test" tab, adjust the "Fail condition" and "Violation tolerance".
4. Define the eye mask. See Chapter 10.3.2, "Eye mask definition", on page 407.

The general mask test settings are described in Chapter 10.2, "Mask test settings", on page 395.

10.3.2 Eye mask definition

Access: [App Cockpit] > "Analysis" tab > "Define Mask" > "Mask Type" = "Eye" > "Setup mask"

The "Eye Mask " dialog provides all settings to define masks of different shapes. It is available if option R&S RTO-K12 is activated on the instrument.



Shape
Defines the outline of the eye mask: square, diamond, hexagon or octagon.

Remote command:
MTESt:EYEMask:TYPE on page 1597

Horizontal center/Vertical center
Set the horizontal (time) and vertical (voltage) values of the eye shape enter and thus define the position of the eye shape on the display.

Remote command:
MTESt:EYEMask:HPOSition on page 1600
MTESt:EYEMask:VPOSition on page 1600

Main width (W1), Minor width (W2)
Main width defines the width of all eye mask shapes. Minor width defines the secondary width for hexagon and octagon mask shapes.

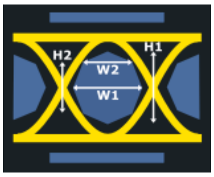


Figure 10-6: Main and minor widths and heights of an octagon eye mask

Remote command:
MTESt:EYEMask:WIDTh<m>[:VALue] on page 1598

Main height (H1), Minor height (H2)
Main height defines the height of all eye mask shapes. Minor height defines the secondary height for octagon mask shapes.

If an octagon shape is selected, and both values are the same, the eye masks looks like a hexagon rather than an octagon.

Remote command:
MTESt:EYEMask:HEIGht<m>[:VALue] on page 1598

Periodicity
Sets the time distance between the shape centers.

Remote command:
MTESt:EYEMask:HPERiod on page 1599

Top/Bottom setup
Defines the settings for the top and bottom segments of the mask. You have to enable "Top" / "Bottom" for this settings to be relevant.

Symmetric ← Top/Bottom setup
Sets bottom and top offsets to the same value so that the outer regions are symmetric to the eye shape.

Remote command:
MTESt:EYEMask:TBSYmmetric on page 1600

Width ← Top/Bottom setup
Sets the time width of the outer regions, symmetric to the eye shape center.

Remote command:
MTESt:EYEMask:TBWidth on page 1600

Offset top, Offset bottom ← Top/Bottom setup
Sets the voltage distance from the eye shape center that limits the upper and lower regions.

Remote command:
MTESt:EYEMask:BOFFset on page 1599
MTESt:EYEMask:TOFFset on page 1599

Segments
Enable the different regions of the mask: "Top", "Bottom", "Left", "Right".

The icons copy the eye shape to the left and to the right.

Remote command:
`MTESt:EYEMask:MSKBottom` on page 1599
`MTESt:EYEMask:MSKTop` on page 1599
`MTESt:EYEMask:MSKLeft` on page 1598
`MTESt:EYEMask:MSKRight` on page 1598

Convert to user mask
Converts the test and mask definitions of the current mask test to a new mask test of type "User".

The new test is opened in a separate mask test tab, in the "Define Mask" tab.

This is useful, for example, if want to run variants of a mask test in parallel.

Remote command:
`MTESt:CEMask` on page 1597

10.4 Working with masks

This chapter explains step-by-step how mask tests are setup and preformed. For the explanation of the individual settings, see Chapter 10.2, "Mask test settings", on page 395.

10.4.1 Setting up user masks

10.4.1.1 Creating user masks

There are two ways to create a new mask:

- Graphical way by tapping the mask points on the touchscreen,
- Numerical entry of the x- and y-values of the mask points.

You can combine both methods. For example, at first you enter the mask quickly on the touchscreen, and then modify the point coordinates with precise values.

To create a mask graphically on the touch screen

1. Tap the "Masks" icon on the toolbar.

The "Masks" toolbar assist opens.

2. Tap the corner points of the mask segment on the touch screen.

   Tip: To create an exact rectangle, draw the diagonal of the rectangle on the screen.

3. Tap "Finish segment".

   Now you can enter another segment to the current mask test or add a new mask test.

4. To finish the mask definition, close the toolbar assist.
   If the acquisition is running, the mask test starts automatically and immediately.

To create a mask numerically in the dialog box

The settings mentioned here are described in detail in Chapter 10.2.1, "Define mask", on page 395.

1. Open [App Cockpit] > "Analysis" tab > "Mask" > "Define Mask" tab.

2. Create a mask test:

   a) Tap the "+"-icon in the upper right corner.
   b) Enter a name for the new mask test.

   A new, empty tab for the mask test appears.

3. Set "Mask type"= "User".

4. Tap "Setup mask".

5. In the "User Mask: Segments" area, tap "Insert" to create a new mask segment.

6. Set the corner points of the mask segment:

   a) Select the "Segment".
   b) Tap "Configure". The "User Mask: Points" dialog opens.
   c) In the "Definition of segment" area, tap "Insert".
      Point 1 appears.
   d) Tap the X-cell and enter the X-value of the point.
   e) Tap the Y-cell and enter the Y-value of the point.
   f) To insert the next point:
      - Tap "Insert" to add a point before the selected point.
      - Tap "Append" to add a point at the end of the list.
   g) Set the X- and Y-values for this point.
   h) Repeat the last two steps until you define all points.

7. In the "Define Mask" tab, tap "Enable".

8. Adjust the horizontal and vertical units if necessary.

10.4.1.2 Modifying user masks

To change an existing mask definition, you can also use the graphical method on the touch screen, or the numerical way, or combine both.

With the graphical method, you can:

- Move, add, and delete segments
- Move and delete points

Adding points to an existing segment graphically is not possible.

With the numerical method, in the "Define Mask" tab, you have all modification possibilities. You can delete and add points and segments, change the coordinates, and also stretch a segment, or move it by adding an offset.

To add a mask segment on the touch screen

1. Tap a mask segment of the mask test that you want to complement.
2. Tap the "Masks" icon on the toolbar.

3. Tap the corner points of the new mask segment on the touch screen.
4. To finish the segment and mask definition, double-tap the last point.

To delete a mask segment on the touch screen

1. On the toolbar, tap the "Delete" icon.

2. Tap the mask segment you want to delete.

To delete a point on the touch screen

1. Tap the mask segment from which you want to delete a point.

   The selected segment is now in definition mode, shown with blue color.
2. On the toolbar, tap the "Delete" icon.

   The "Delete" toolbar assist opens.
3. Tap the point you want to delete.

To move a segment on the touch screen

1. Drag&drop the segment to the new position.
2. Tap outside the mask to deselect the mask segment.

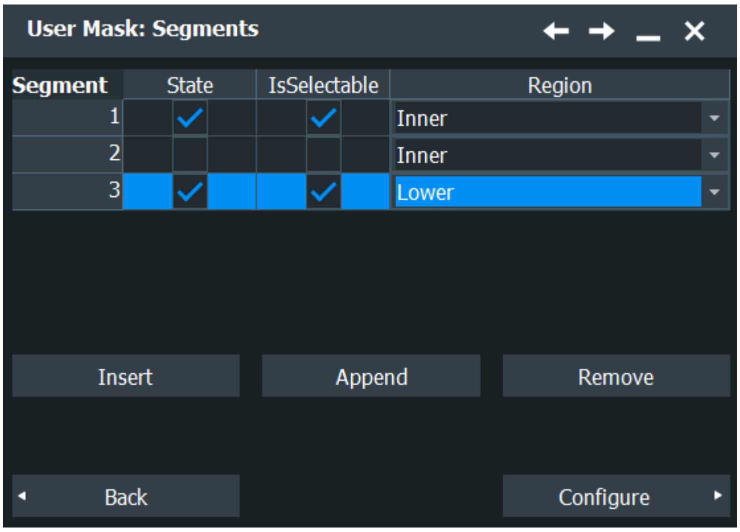
To move a point on the touch screen

1. Tap the mask segment that you want to change.
2. Drag&drop the point to the new position.
3. Tap outside the mask to deselect the mask segment.

To change the mask definition numerically

The settings mentioned here are described in detail in Chapter 10.2.2, "Mask definition", on page 397.

1. Open [App Cockpit] > "Analysis" tab > "Mask" > "Define Mask".
2. At the top, select the mask test for which you want to change the mask definition.
3. To add or delete a mask segment, tap "Setup mask".
   Tap the segment's row in the "User Mask: Segments" table and tap the required button below:
   - "Insert": to add a new segment before the selected segment.
   - "Append": to add a new segment at the end of the list.
   - "Remove": to delete the selected mask segment from the list.

4. To add, delete, or move a point of a segment:
   a) Select the segment in the "User Mask: Segments" table.
   b) Tap "Configure".
   c) Select the point in the "Definition of segment" table.

d) To add or delete the selected point, use the buttons below the table.
   - "Insert": to add a new point before the selected point.
   - "Append": to add a new point at the end of the list.
   - "Remove": to delete the selected point from the list.

e) To move the selected point, change the X- and Y-values.

To rescale and move a mask segment

The settings mentioned here are described in detail in Chapter 10.2.2, "Mask definition", on page 397.

1. Open [App Cockpit] > "Analysis" tab > "Mask" > "Define Mask".
2. At the top, select the mask test for which you want to change the mask definition.
3. Tap "Setup mask" and select the required segment in the "User Mask: Segments" table.
4. Tap "Configure".
5. To stretch or compress the selected mask segment, enter the "Factor X" for horizontal scaling and the "Factor Y" for vertical scaling. The x-values and y-values of all points are multiplied with the corresponding factor. Factors >1 stretch the mask segment, while factors between 0 and 1 compress it. Negative values are possible and change the algebraic sign.

6. To move the selected mask segment, enter the "Offset X" for horizontal direction and the "Offset Y" for vertical direction. The specified offset is added to the corresponding values of all points.
7. Tap "Apply" to perform the scaling and/or move.

10.4.2 Setting up a mask test

In addition to the mask definition, the mask test contains further settings:

- The waveform to be tested
- The criteria for a failed test
- The actions to be taken if a test has failed or has been completed successfully

1. Open [App Cockpit] > "Analysis" tab > "Mask" > "Define Mask" tab.
2. Select the "Target" to be tested.
3. Select the "Define Test" tab.
4. Set the conditions for a failed test:
   a) " Fail condition": select if sample hits or the number of acquisitions with sample hits are considered.
   b) "Violation tolerance": number of tolerable sample hits or acquisition hits.

   A test has failed if the number of sample hits or acquisition hits exceeds the limit of violation tolerance hits.
5. Tap "Action on mask".
6. For each action, select when the action will be executed:
   - "On violation" if the mask test has failed
   - "On successful completion"

10.4.3 Configuring the mask and hit display

The display of masks and mask violation is the same for all mask tests .

The settings mentioned here are described in detail in Chapter 10.2.3, "Actions on event", on page 404.

1. Open [App Cockpit] > "Analysis" tab > "Mask".
2. Select the "Display" tab.
3. Select "Show mask" to display the masks of all enabled mask tests on the screen.
4. Open "Menu" > "Settings" >"Appearance" dialog.
5. Select the "Mask" tab.
6. Define how the sample hits are displayed:

a) Select "Show hits" to display the sample hits.
b) Set the "Show for" time or "Show hits forever".
c) Set the "Color" of the sample hits.

7. Define the color of the masks segments depending on the violation state:
   - Mask without violation
   - Mask with violation
   - Mask with contact: this color shows that the edge of the mask segment was touched. In this case, the resolution is not sufficient to detect if the mask was really hit or not. Zoom into the concerned area to see the correct result.

10.4.4 Running a mask test

Before you can start a mask test, make sure that the mask setup is complete:
- The mask is defined, see Chapter 10.4.1.1, "Creating user masks", on page 409 and Chapter 10.4.1.2, "Modifying user masks", on page 411.
- The mask test is defined, see Chapter 10.4.2, "Setting up a mask test", on page 414
- The mask display is configured, see Chapter 10.4.3, "Configuring the mask and hit display", on page 414.

You can perform continuous testing or test a specified number of acquisitions.

1. Open [App Cockpit] > "Analysis" tab > "Mask" > "Define Mask" tab.
2. Select "Enable".
   If the acquisition is running, the test starts immediately.
3. If the acquisition is not running, press [Run Stop].
   The test starts and runs until you stop the acquisition or the stop action is executed if defined.

10.4.5 Saving and loading masks

Mask test definitions remain on the instrument until they are changed or deleted, or [Preset] is performed. If you want to keep a mask test, you can save and reload them.

To save a mask

1. Open [App Cockpit] > "Analysis" tab > "Mask" > "Define Mask" tab.
2. To save the mask file in the current directory, change the file name if needed, and tap "Save".
   You can use the automatic file name generation, see Chapter 4.8.1, "Autonaming", on page 110.
3. To select the directory and enter the file name, tap "Save As".

To load a mask

1. Tap "Open".
2. Select the file from the file selection dialog box.

10.4.6 Mask testing on history acquisitions

In the same way as for running acquisitions, you can set up and perform the mask testing also on history waveforms.

The requirements for mask testing on history waveforms are also the same:

- The mask is defined, see Chapter 10.4.1.1, "Creating user masks", on page 409 and Chapter 10.4.1.2, "Modifying user masks", on page 411.
- The mask test is defined, see Chapter 10.4.2, "Setting up a mask test", on page 414
- The mask display is configured, see Chapter 10.4.3, "Configuring the mask and hit display", on page 414.

1. Perform and finish the acquisition.
2. Press [History].
3. In the quick-access "History" dialog box, tap "Play".

   The mask testing is performed on the complete history memory, starting with the oldest acquisition. The state of the mask test is set to "Finished" when "Nx Single count" acquisitions are tested.

For details on history, see Chapter 7.4.1, "About history", on page 278.

11 Search functions

Search functions allow you to detect and analyze specific events in the acquired data quickly and simply. You can search in various waveforms for several events at once. The search area can be limited by a gate.

The events that can be searched for are basically the same events you can trigger on. Thus, the search parameters are defined in the same way as the trigger conditions. The results are displayed in a result box and optionally shown in a zoom window.

11.1 Overview: search definition and results

11.1.1 Search definition

You can define up to 8 different searches and let them run simultaneously. For each search, you define the criteria, the parameters of each criterion, the gate, and the result display.

The instrument keeps the settings until the next preset. If you save a user-defined preset, the search settings are included in the preset.

Each search is configured in a separate tab and contains:

- *Search control*
  If you enable a search and run an acquisition, the search is performed continuously on the acquired data until acquisition is stopped.
  If acquisition is stopped and you enable a search, the data of the last acquisition is searched.
  Enabling the search zoom window disables the search, stops a running acquisition, and displays the search results of the last acquisition in the zoom window.
- *Source*
  Waveform that is searched for one or more events. You can search in analog and digital signals, math or reference waveforms, and tracks. Furthermore, search in decoded data of serial buses is possible.
- *Search criteria and parameters*
  Various search criteria are available, depending on the source. Most parameters known from trigger event definition can also be configured as search conditions.
  Unlike triggering, you can configure several event types to be searched for simultaneously.
  If the source is an FFT spectrum, you can perform a frequency marker search by using the cursor measurement and defining the peak excursion. See Chapter 8.1.3.3, "Peak search tab", on page 297.
- *Search gate*
  Searches can be performed on the entire waveform, or only on a defined area (gate). The gate can be coupled to an existing zoom.
  Gating is not available for searches on digital signals and serial buses.

- *Result presentation*
  For each search, you define how the search results are displayed: in a result table and/or in a search zoom window.

Remote commands:
- `SEARch:ADD` on page 1602
- `SEARch:REMove` on page 1602

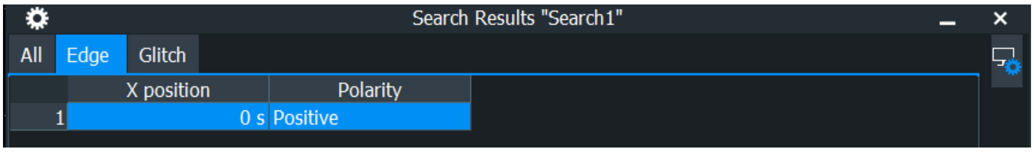
11.1.2 Search results

The results are displayed in a "Search Results" box and optionally in a zoom window.

Search Results box

The results of each search are tabulated in a "Search Results" box.

If you search for several event types in parallel, the results are presented in several tabs - one for each search event and one for the combined results. Each tab contains a table with the position and, if available, further parameters for each result. The tables row can be sorted, and you can define a maximum number of table entries in the "Result Presentation" dialog box. As with all result boxes, you can minimize it, display it like a diagram, and define the default position.



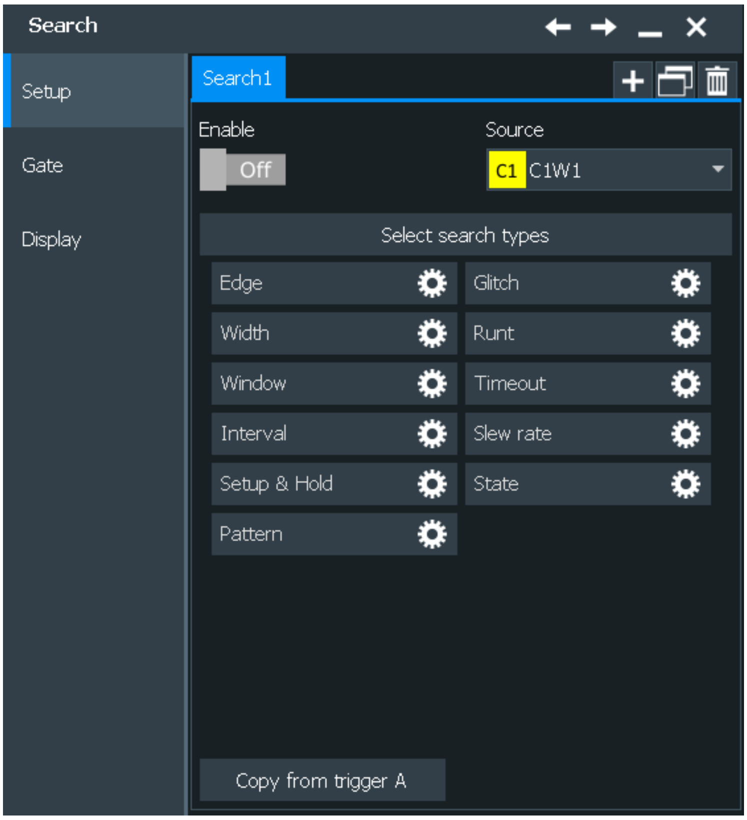
If "Auto clear" is enabled in the "Display" dialog box, the instrument displays the search results of the last acquisition. If "Auto clear" is disabled, the first result of each acquisition is listed until the maximum number of entries in the table is reached.

Remote commands for result query:
- `SEARch:RESult[:ALL]?` on page 1637

11.2 Search setup

Access: "Menu" > "Apps" > "Analysis" tab > "Search"

The search setup includes the source selection, the selection of search events (criteria), event-specific search conditions, and search control.



11.2.1 Search criteria

Access: "Menu" > "Apps" > "Analysis" tab > "Search" > "Setup" tab

Enable

If you enable a search and run an acquisition, the search is performed continuously on the acquired data until acquisition is stopped.

If acquisition is stopped and you enable a search, the data of the last acquisition is searched.

Remote command:
`SEARch:ONLine` on page 1603
`SEARch:ALL` on page 1603

Source

Defines the waveform to be searched. The source can be any analog and digital input signal, math or reference waveform, or track. While the instrument triggers only on real input signals, it can search also calculated and restored waveforms.

If the source is an FFT spectrum, you can perform a frequency marker search by using the cursor measurement and defining the peak excursion. See Chapter 8.1.3.3, "Peak search tab", on page 297.

For some serial protocol options, search on a serial bus is available. For details, see the relevant chapters of the "Protocol Analysis" chapter.

Depending on the selected source, different search criteria are available.

Remote command:
SEARch:SOURce on page 1602

Edge, Glitch, Width, Runt, Window, Timeout, Interval, Slew rate, Setup & Hold, State, Pattern
Search criteria for analog and digital input signals, math and reference waveforms, and tracks. For searching on digital channels, only "Edge", "Width", "Timeout" and "Setup & Hold" criteria are available.

Tap the icon to include or exclude the search criteria in the next search. You can enable several event types for simultaneous search.

Remote command:
SEARch:TRIGger:EDGE[:STATe] on page 1604
SEARch:TRIGger:GLITch[:STATe] on page 1604
SEARch:TRIGger:WIDTh[:STATe] on page 1604
SEARch:TRIGger:RUNT[:STATe] on page 1604
SEARch:TRIGger:WINDow[:STATe] on page 1604
SEARch:TRIGger:TIMeout[:STATe] on page 1604
SEARch:TRIGger:INTerval[:STATe] on page 1604
SEARch:TRIGger:SLEWrate[:STATe] on page 1604
SEARch:TRIGger:SETHold[:STATe] on page 1604
SEARch:TRIGger:STATe[:STATe] on page 1604
SEARch:TRIGger:PATTern[:STATe] on page 1604

Copy from trigger A'
Copies the trigger type-specific settings from the A-trigger configuration to the search settings. The source itself is not copied.

Remote command:
SEARch:TRIGger:EDGE:ACOPy on page 1605
SEARch:TRIGger:GLITch:ACOPy on page 1605
SEARch:TRIGger:WINDow:ACOPy on page 1605
SEARch:TRIGger:WIDTh:ACOPy on page 1605
SEARch:TRIGger:RUNT:ACOPy on page 1605
SEARch:TRIGger:WINDow:ACOPy on page 1605
SEARch:TRIGger:TIMeout:ACOPy on page 1605
SEARch:TRIGger:INTerval:ACOPy on page 1605
SEARch:TRIGger:SLEWrate:ACOPy on page 1605
SEARch:TRIGger:SETHold:ACOPy on page 1605
SEARch:TRIGger:STATe:ACOPy on page 1605
SEARch:TRIGger:PATTern:ACOPy on page 1605

11.2.2 Search parameters

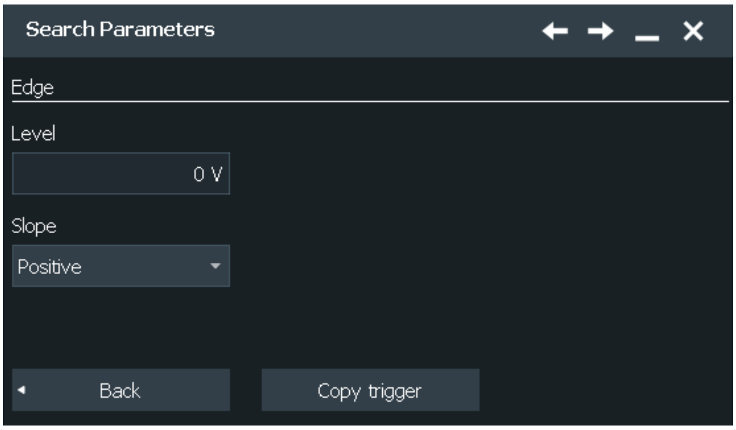
Most parameters available for trigger event definition can also be configured as search conditions. Each event type is defined in a separate subtab.

If the source is a spectrum, the instrument performs a frequency marker search.

For serial protocol options, search on a serial bus is available. These searches have protocol-specific search criteria. For details, see the relevant chapters of the "Protocol Analysis" chapter.

11.2.2.1 Edge

The edge search works the same way as the edge trigger.



Slope

Sets the edge type: rising edge ("Positive"), falling edge ("Negative"), or both.

Remote command:
SEARch:TRIGger:EDGE:SLOPe on page 1606

Level

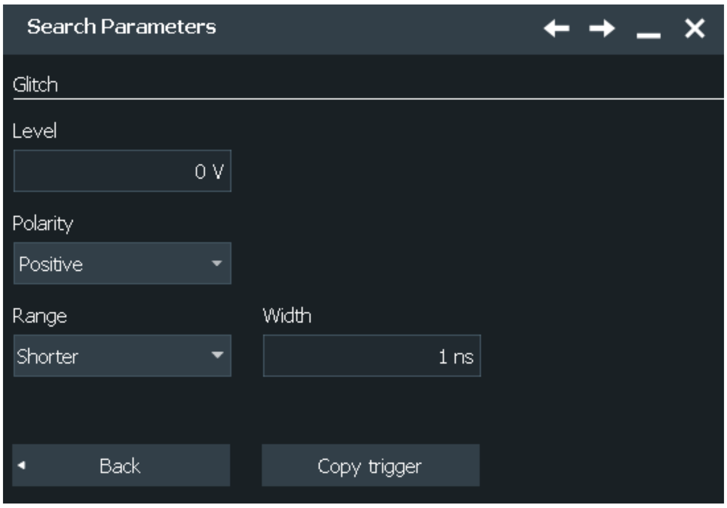
Sets the voltage level for trigger search. The value is used for all search event types that require one trigger level. The search trigger level is search-specific, you can define different levels in different searches for the same event.

Remote command:
SEARch:TRIGger:LEVel[:VALue] on page 1605

11.2.2.2 Glitch

The glitch search works the same way as the glitch trigger. To apply the trigger settings to search, tap Copy from trigger A'.

The glitch search is not available if the search source is a digital channel.



Polarity, Range, Width
See trigger settings:
- "Range" on page 201
- "Width" on page 201
- "Polarity" on page 201

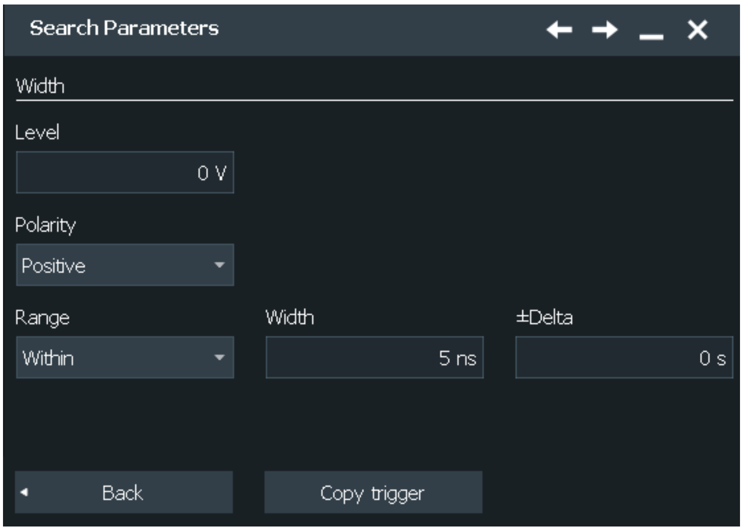
Remote command:
`SEARch:TRIGger:GLITch:POLarity` on page 1606
`SEARch:TRIGger:GLITch:RANGe` on page 1607
`SEARch:TRIGger:GLITch:WIDTh` on page 1607

Level
Sets the voltage level for trigger search. The value is used for all search event types that require one trigger level. The search trigger level is search-specific, you can define different levels in different searches for the same event.

Remote command:
`SEARch:TRIGger:LEVel[:VALue]` on page 1605

11.2.2.3 Width

The width search works the same way as the width trigger. To apply the trigger settings to search, tap Copy from trigger A'.



Polarity, Range, Width, ±Delta

See trigger settings:

- "Polarity" on page 202
  While the width trigger can only analyze positive *or* negative polarity, searching for a width is also possible for both polarities at the same time ("Either").
- "Range" on page 202
- "Width" on page 203
- "±Delta" on page 203

Remote command:
`SEARch:TRIGger:WIDTh:POLarity` on page 1615
`SEARch:TRIGger:WIDTh:RANGe` on page 1615
`SEARch:TRIGger:WIDTh:WIDTh` on page 1616
`SEARch:TRIGger:WIDTh:DELTa` on page 1615

Level

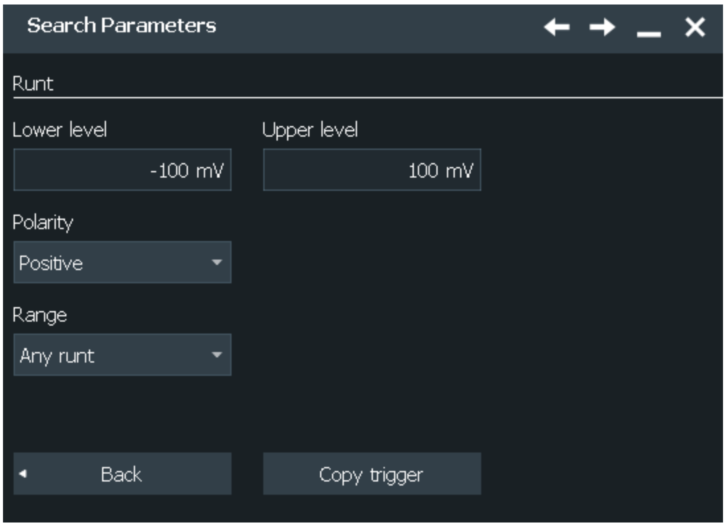
Sets the voltage level for trigger search. The value is used for all search event types that require one trigger level. The search trigger level is search-specific, you can define different levels in different searches for the same event.

Remote command:
`SEARch:TRIGger:LEVel[:VALue]` on page 1605

11.2.2.4 Runt

The runt search settings are the same as the runt trigger settings. To apply the trigger settings to search, tap Copy from trigger A'.

The runt search is not available, if the search source is a digital channel.



Polarity, Range, Runt width, ±Delta
Time limit for the runt, see trigger settings:
- "Polarity" on page 201
- "Range" on page 204
- "Runt width" on page 204
- "±Delta" on page 204

Remote command:
SEARch:TRIGger:RUNT:POLarity on page 1609
SEARch:TRIGger:RUNT:RANGe on page 1610
SEARch:TRIGger:RUNT:WIDTh on page 1610
SEARch:TRIGger:RUNT:DELTa on page 1609

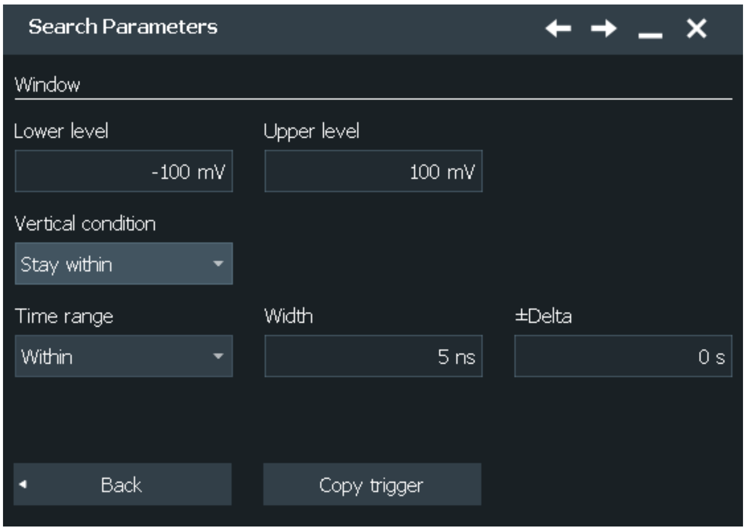
Upper level, Lower level
Set the upper and lower voltage thresholds. The amplitude of a runt crosses the first threshold twice in succession without crossing the second one.

Remote command:
SEARch:TRIGger:LEVel:RUNT:UPPer on page 1611
SEARch:TRIGger:LEVel:RUNT:LOWer on page 1611

11.2.2.5 Window

The window search settings are the same as the window trigger settings. This search type is not available if the search source is a digital channel. To apply the trigger settings to search, tap Copy from trigger A'.

The window search is not available if the search source is a digital channel.



Vertical condition

Defines the run of the signal relative to the window, see "Vertical condition" on page 205.

Remote command:
`SEARch:TRIGger:WINDow:RANGe` on page 1617

Time condition, Width, ±Delta

Set the time limit for the vertical condition, see:

- "Time condition" on page 206
- "Width" on page 206
- "±Delta" on page 206

Remote command:
`SEARch:TRIGger:WINDow:TIMerange` on page 1618
`SEARch:TRIGger:WINDow:WIDTh` on page 1618
`SEARch:TRIGger:WINDow:DELTa` on page 1617

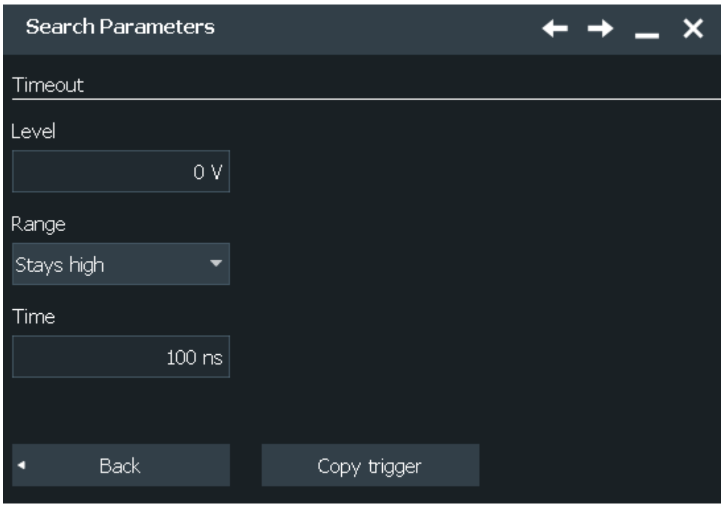
Upper level, Lower level

Set the upper and lower voltage thresholds. The amplitude of a runt crosses the first threshold twice in succession without crossing the second one.

Remote command:
`SEARch:TRIGger:LEVel:WINDow:UPPer` on page 1619
`SEARch:TRIGger:LEVel:WINDow:LOWer` on page 1619

11.2.2.6 Timeout

The timeout search settings are the same as the timeout trigger settings. To apply the trigger settings to search, tap Copy from trigger A'.



Range, Time

Set the timeout condition, see:

- "Range" on page 207
- "Time" on page 207

Remote command:

SEARch:TRIGger:TIMeout:RANGe on page 1614

SEARch:TRIGger:TIMeout:TIME on page 1614

Level

Sets the voltage level for trigger search. The value is used for all search event types that require one trigger level. The search trigger level is search-specific, you can define different levels in different searches for the same event.

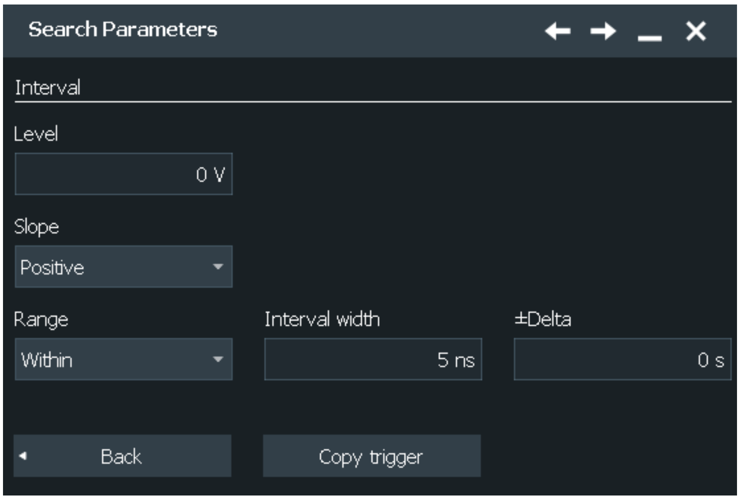
Remote command:

SEARch:TRIGger:LEVel[:VALue] on page 1605

11.2.2.7 Interval

The interval search settings are the same as the interval trigger settings. To apply the trigger settings to search, tap Copy from trigger A'.

The interval search is not available if the search source is a digital channel.



Slope, Range, Interv. width, ±Delta

Set the interval condition, see:

- "Slope" on page 207
  While the interval trigger can only analyze rising or falling edges, searching for a width is possible for both edges at the same time ("Either").
- "Range" on page 208
- "Interv. width" on page 208
- "±Delta" on page 208

Remote command:
`SEARch:TRIGger:INTerval:SLOPe` on page 1607
`SEARch:TRIGger:INTerval:RANGe` on page 1608
`SEARch:TRIGger:INTerval:WIDTh` on page 1609
`SEARch:TRIGger:INTerval:DELTa` on page 1608

Level

Sets the voltage level for trigger search. The value is used for all search event types that require one trigger level. The search trigger level is search-specific, you can define different levels in different searches for the same event.

Remote command:
`SEARch:TRIGger:LEVel[:VALue]` on page 1605

11.2.2.8 Slew rate

The slew rate search settings are the same as the slew rate trigger settings. To apply the trigger settings to search, tap Copy from trigger A'.

The slew rate search is not available if the search source is a digital channel.

Polarity, Range, Slew rate, ±Delta

- "Slope" on page 200
- "Range" on page 209
- "Slew rate" on page 210
- "±Delta" on page 210

Remote command:
SEARch:TRIGger:SLEWrate:SLOPe on page 1612
SEARch:TRIGger:SLEWrate:RANGe on page 1612
SEARch:TRIGger:SLEWrate:TIME on page 1613
SEARch:TRIGger:SLEWrate:DELTa on page 1611

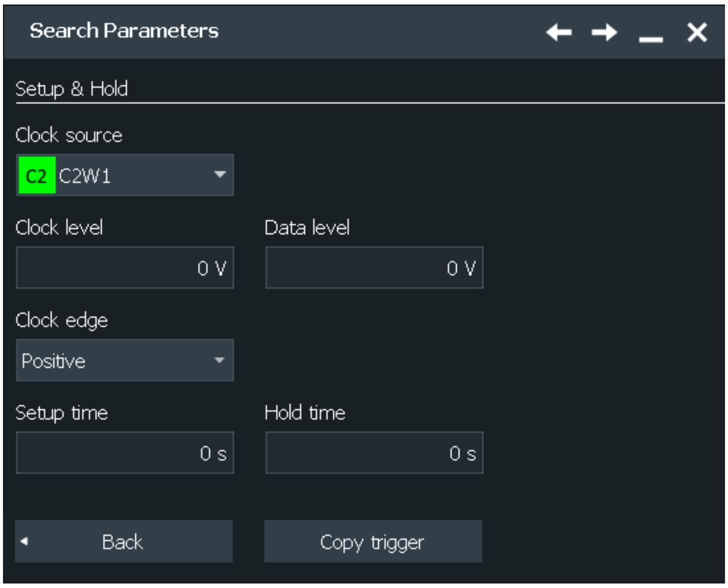
Upper level, Lower level

Set the upper and lower voltage thresholds. When the signal crosses a level, the slew rate measurement starts or stops depending on the selected slope.

Remote command:
SEARch:TRIGger:LEVel:TRANsition:UPPer on page 1613
SEARch:TRIGger:LEVel:TRANsition:LOWer on page 1613

11.2.2.9 Setup & Hold

The Setup & Hold search settings are the same as the Setup & Hold trigger settings. To apply the trigger settings to search, tap Copy from trigger A'.



Clock source, Clock edge, Clock level

Set the clock settings. Both "Clock level" and "Clock edge" define the starting point for calculation of the setup and hold time.

Remote command:
SEARch:TRIGger:SETHold:CSOurce on page 1620
SEARch:TRIGger:SETHold:CEDGe on page 1619
SEARch:TRIGger:SETHold:CLEVel on page 1620

Data level

Sets the voltage level for the data signal. At this level, the setup and hold time are measured.

Remote command:
SEARch:TRIGger:LEVel[:VALue] on page 1605

Level

Sets the voltage level for trigger search. The value is used for all search event types that require one trigger level. The search trigger level is search-specific, you can define different levels in different searches for the same event.

Remote command:
SEARch:TRIGger:LEVel[:VALue] on page 1605

Setup time, Hold time

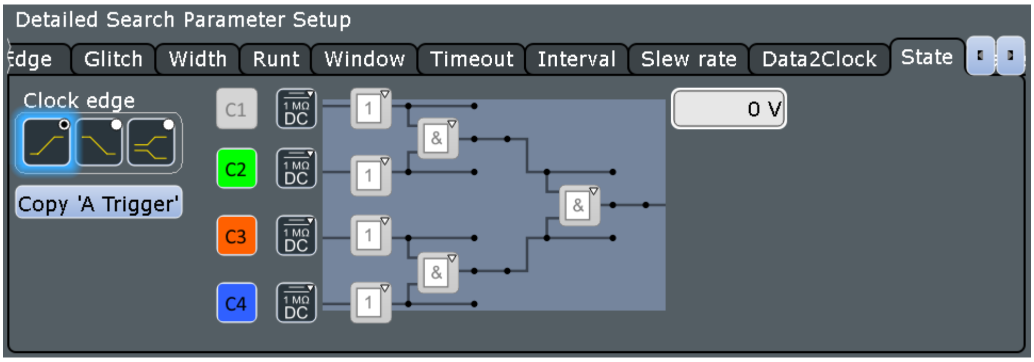
Sets the minimum time **before** (Setup) and **after** (Hold) the clock edge while the data signal must stay steady above or below the data level.

See also: "Setup time" on page 212 and "Hold time" on page 212.

Remote command:
SEARch:TRIGger:SETHold:STIMe on page 1621
SEARch:TRIGger:SETHold:HTIMe on page 1620

11.2.2.10 State

The state search is a qualified edge search. The state search is only available for analog channel sources (Ch1 to Ch4).



Clock source, Clock edge
Define the clock settings. The clock signal is the waveform to be searched.

Remote command:
SEARch:TRIGger:STATe:CSOurce on page 1626
SEARch:TRIGger:STATe:CEDGe on page 1626
SEARch:TRIGger:STATe:CLEVel on page 1626

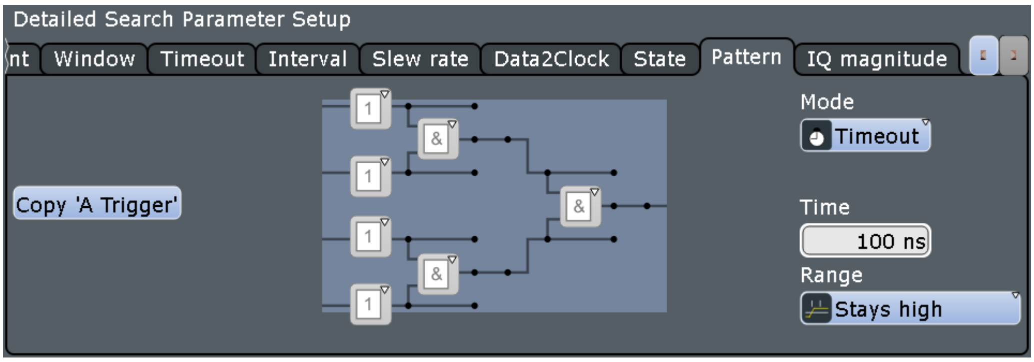
State pattern
State settings are the same as for the state trigger.

See also "Pattern" on page 213.

Remote command:
SEARch:TRIGger:STATe:A[:ENABle] on page 1627
SEARch:TRIGger:STATe:A:LOGic on page 1627
SEARch:TRIGger:STATe:AB:LOGic on page 1628
SEARch:TRIGger:STATe:ABCD:LOGic on page 1628

11.2.2.11 Pattern

The pattern search combines a logical combination of the input channels with a timing condition. The pattern search is only available for analog channel sources (Ch1 to Ch4).



Pattern
Pattern search settings are the same as for the pattern trigger.

See also "Pattern" on page 213.

Remote command:
SEARch:TRIGger:PATTern:A[:ENABle] on page 1622
SEARch:TRIGger:PATTern:A:LOGic on page 1622
SEARch:TRIGger:PATTern:AB:LOGic on page 1623
SEARch:TRIGger:PATTern:ABCD:LOGic on page 1623

Timing condition: Mode, Range, Time, Width, ±Delta
Additional time limitation to the pattern, see "Additional settings: Timing" on page 215.

Remote command:
SEARch:TRIGger:PATTern:MODE on page 1623
SEARch:TRIGger:PATTern:TIMeout:MODE on page 1624
SEARch:TRIGger:PATTern:TIMeout[:TIME] on page 1624
SEARch:TRIGger:PATTern:WIDTh:RANGe on page 1624
SEARch:TRIGger:PATTern:WIDTh[:WIDTh] on page 1625
SEARch:TRIGger:PATTern:WIDTh:DELTa on page 1625

11.2.3 Frequency marker search

When you start a search on a spectrum, a frequency marker search is performed to detect peaks in a spectrum. You can define which peaks the instrument finds by defining the noise reject settings.

Threshold
See "Threshold" on page 298.

Peak excursion
See "Peak excursion" on page 298.

11.2.4 Configuring the search setup

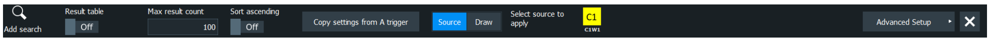
There are several ways to create a search:

- Creating a simple default search using the toolbar icon. This method is not available for search on serial buses.
- Setting up a search using the dialog box.

To perform a simple search

1. If more than one waveform is in the diagram, select the waveform to be searched for by tapping it in the diagram.
2. Select the "Search" icon on the toolbar.

The toolbar assist opens.



3. Tap the diagram with the waveform to be searched, or drag a rectangle on the diagram to define the search area.
4. Alternatively, use the settings provided in the toolbar assist.

   The default edge search is configured as "Search<x>" and performed. The "Search Results" box is displayed.

To create a user-defined search

1. Open the search dialog: "Menu" > "Apps" > "Analysis" tab > "Search".
2. There are two ways to create a search:
   - If you want to create a new, unconfigured search, tap the "Add" icon.

     
   - If you want to create a new search based on an existing one, tap the "Copy" icon.

     
3. Enter a name for the search using the on-screen keyboard.

To configure a user-defined search

1. Open the search dialog: "Menu" > "Apps" > "Analysis" tab > "Search".
2. Select the "Setup" tab and the search you want to configure.
3. Select the "Source" on which you want to perform the search.
4. Select the events you want to include in the search.
5. Define the settings of the first search event.
   To use the same conditions as defined in the trigger configuration of the A-event, tap "Copy from trigger A'". The selected trigger settings are applied to the search settings.
6. Repeat the previous steps to define further events for the same search.
7. To perform the search only on a part of the waveform, configure the gate in the "Gate" tab as described in Chapter 11.3.2, "Defining the search gate", on page 435.

11.3 Search gate

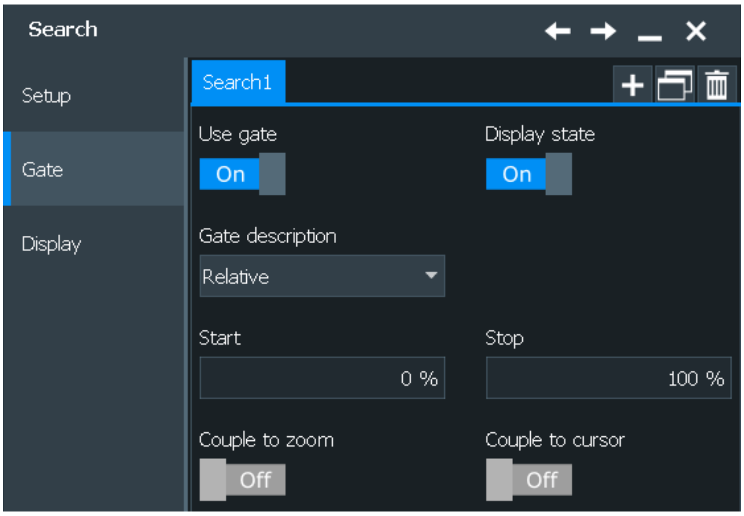
The gate defines the search area within the source waveform. You can use absolute or relative values to define the gate, or couple it to a previously defined zoom area.

11.3.1 Gate settings

Access: "Menu" > "Apps" > "Analysis" tab > "Search" > "Gate" tab

The search gate settings are identical to those for gate areas for measurements or FFT analysis.

Gating is not available if the search source is a digital channel or a serial bus.



Use gate
Enables the gate settings and shows the gate. Search is only performed on the defined gate area of the source waveform.

Remote command:
`SEARch:GATE[:STATe]` on page 1629

Display state
Displays the gate area in the source diagram.

Remote command:
`SEARch:GATE:SHOW` on page 1630

Gate description
Defines whether the gate settings are configured using absolute or relative values.

"Absolute"	The gate is defined by absolute start and stop values.
"Relative"	The gate's start and stop values are defined by a percentage of the value range.

Remote command:
`CALCulate:MATH<m>:FFT:GATE:MODE` on page 1571
`MEASurement<m>:GATE:MODE` on page 1545
`SEARch:GATE:MODE` on page 1629

(Relative) Start
Defines the starting value for the gate.

Remote command:
`CALCulate:MATH<m>:FFT:GATE:ABSolute:STARt` on page 1571
`CALCulate:MATH<m>:FFT:GATE:RELative:STARt` on page 1572
`MEASurement<m>:GATE:ABSolute:STARt` on page 1545
`MEASurement<m>:GATE:RELative:STARt` on page 1545
`SEARch:GATE:ABSolute:STARt` on page 1630
`SEARch:GATE:RELative:STARt` on page 1630

(Relative) Stop
Defines the end value for the gate.

Remote command:
`CALCulate:MATH<m>:FFT:GATE:ABSolute:STOP` on page 1571
`CALCulate:MATH<m>:FFT:GATE:RELative:STOP` on page 1572
`MEASurement<m>:GATE:ABSolute:STOP` on page 1545
`MEASurement<m>:GATE:RELative:STOP` on page 1545
`SEARch:GATE:ABSolute:STOP` on page 1630
`SEARch:GATE:RELative:STOP` on page 1631

Zoom
Zoom coupling is available if a zoom is defined. As long as "Zoom coupling" is enabled, the gate area is defined identically to the zoom area - if you change the zoom, the gate changes as well.

If several zoom diagrams are defined, select the zoom diagram to be used for gating. The "Start" and "Stop" values of the gate are adjusted accordingly.

Zoom coupling can be set for measurement gates, FFT gates, and search gates. The zoom must be defined on the diagram that contains the signal source of the measurement, FFT, or search.

Remote command:
`MEASurement<m>:GATE:ZCOupling` on page 1546
`MEASurement<m>:GATE:ZDIagram` on page 1547
`CALCulate:MATH<m>:FFT:GATE:ZCOupling` on page 1572
`SEARch:GATE:ZCOupling` on page 1631
`SEARch:GATE:ZDIagram` on page 1631

Cursor
If enabled, the gate area is defined by the cursor lines of an active cursor measurement.

If several cursor measurements are enabled, select the cursor set to be used for gating. The "Start" and "Stop" values of the gate are adjusted to the values of the cursor line positions. The measurement is limited to the part of the waveform between the cursor lines.

Remote command:
`MEASurement<m>:GATE:CCOupling` on page 1546
`MEASurement<m>:GATE:CURSor` on page 1546

11.3.2 Defining the search gate

If you create a search using the "Search" toolbar icon, you can directly define the gate by dragging a rectangle on the diagram. Otherwise, you define the gate in the "Gate" tab of the "Search" dialog box.

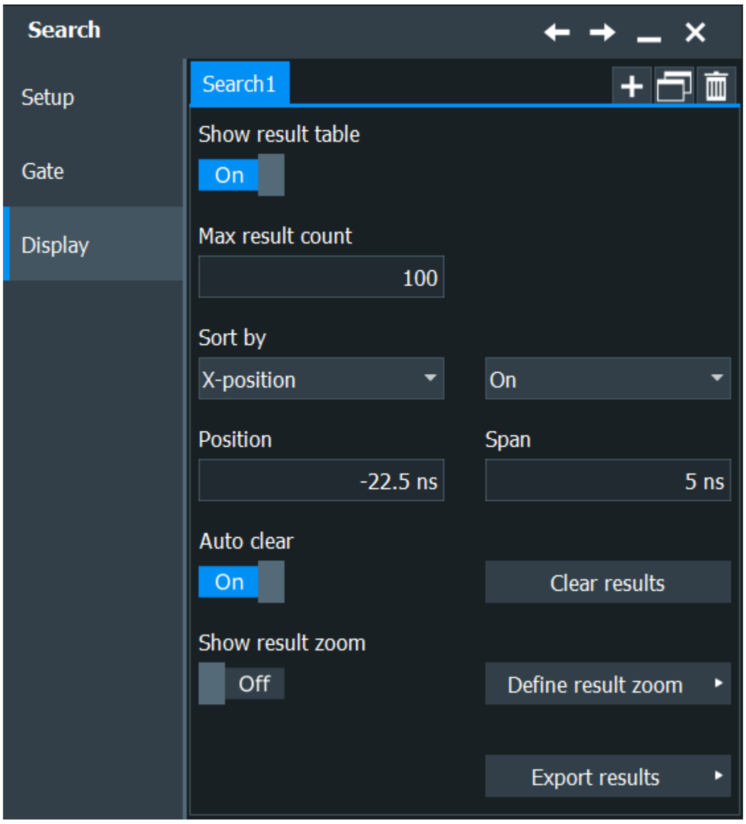
1. Open the "Menu" > "Apps" > "Analysis" tab > "Search" > "Gate" tab.
2. Select the search for which you want to define the gate.
3. Use one of the following methods:
   a) Set the absolute or relative "Gate description" . Enter the start and stop values of the gate area.
   b) If a zoom area has already been defined for the waveform, couple the gate area to the zoom area by selecting the "Couple to zoom" option. If several zoom diagrams are defined, select the zoom diagram you want to use for gating.
4. Tap "Use gate" to enable the gate.
5. Optionally, tap "Display state" to display the gate area in the diagram.

11.4 Search display presentation

Search results are displayed in a table in the "Search Results" box. In addition, a zoom window for a selected search result can be displayed so that you can analyze the result in more detail.

11.4.1 Search display settings

Access: "Menu" > "Apps" > "Analysis" tab > "Search" > "Display"



The following settings configure the layout of the result table in the "Search Results" box and the size and position of the search zoom window. The result tables can be sorted by x-position or value. You can define a maximum number of table entries.

Show result table

Displays or hides the search result table.

Remote command:

`SEARch:RESult:SHOW` on page 1636

Max result count

Defines the maximum number of entries in the search result table.

Remote command:

`SEARch:RESult:LIMit` on page 1635

Sort by

Sorts the search results by x-value position or value of the result.

Remote command:

`SEARch:RESult:SORT[:MODE]` on page 1636

Sort ascending

By default, the results are listed in descending order, i.e. the largest value at the top. To change the sorting direction, enable "Sort ascending".

Remote command:
SEARch:RESult:SORT:ASCending on page 1636

Clear results
Clears the search results once and starts a new search.

Remote command:
SEARch:CLEar on page 1602

Auto clear
If "Auto clear" is enabled, the instrument displays the search results of the last acquisition.

If "Auto clear" is disabled, the first result of each acquisition is listed until the maximum number of entries in the table is reached.

Show result zoom
If enabled, a zoom window is displayed for the currently selected search result. The zoom area is indicated in the diagram that displays the source waveform of the search.

Remote command:
SEARch:RESDiagram:SHOW on page 1634

Define result zoom
Opens a dialog to define the reult zoom display settings.

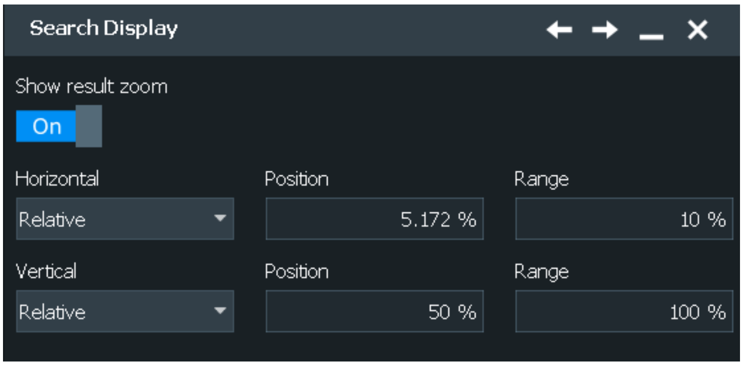
See Chapter 11.4.2, "Display zoom settings", on page 437.

Export results
Opens a dialog to define the export reults.

See Chapter 12.2.4, "Results", on page 464.

11.4.2 Display zoom settings

You can display a zoom window for the currently selected search result. The zoom area is indicated in the diagram that displays the source waveform of the search.



Show result zoom
If enabled, a zoom window is displayed for the currently selected search result. The zoom area is indicated in the diagram that displays the source waveform of the search.

Remote command:
SEARch:RESDiagram:SHOW on page 1634

Y mode

Defines whether absolute or relative values are used to specify the y-axis values.

Remote command:
LAYout:ZOOM:VERTical:MODE on page 1444
SEARch:RESDiagram:VERT:MODE on page 1635

Y position

Defines the y-value of the centerpoint of the zoom area.

Remote command:
LAYout:ZOOM:VERTical:ABSolute:POSition on page 1445
LAYout:ZOOM:VERTical:RELative:POSition on page 1446
SEARch:RESDiagram:VERT:ABSolute:POSition on page 1634
SEARch:RESDiagram:VERT:RELative:POSition on page 1635

Y range

Defines the height of the zoom area.

Remote command:
LAYout:ZOOM:VERTical:RELative:SPAN on page 1447
LAYout:ZOOM:VERTical:ABSolute:SPAN on page 1445
SEARch:RESDiagram:VERT:ABSolute:SPAN on page 1634
SEARch:RESDiagram:VERT:RELative:SPAN on page 1635

X mode

Defines whether absolute or relative values are used to specify the x-axis values.

Remote command:
LAYout:ZOOM:HORZ:MODE on page 1441
SEARch:RESDiagram:HORZ:MODE on page 1633

X position

Defines the x-value of the centerpoint of the zoom area.

Remote command:
LAYout:ZOOM:HORZ:ABSolute:POSition on page 1441
LAYout:ZOOM:HORZ:RELative:POSition on page 1443
SEARch:RESDiagram:HORZ:ABSolute:POSition on page 1632
SEARch:RESDiagram:HORZ:RELative:POSition on page 1633

X range

Defines the width of the zoom area.

Remote command:
LAYout:ZOOM:HORZ:ABSolute:SPAN on page 1442
LAYout:ZOOM:HORZ:RELative:SPAN on page 1443
SEARch:RESDiagram:HORZ:ABSolute:SPAN on page 1633
SEARch:RESDiagram:HORZ:RELative:SPAN on page 1633

11.4.3 Configuring the search results presentation

Initially, the "Search Results" box is displayed in front of the other diagrams or as result icon on the signal bar. This depends on the default setting in the "Diagram Layout" tab. Alternatively, you can display it in its own area on the screen, like any other diagram.

For details, see Chapter 3.3.7, "Displaying results", on page 63.

To configure the result tables

1. Open the search dialog: "Menu" > "Apps" > "Analysis" tab > "Search".
2. Select the tab for the search you want to configure.
3. Select the "Display" tab.
4. Select "Show result table" to display the "Search Results" box.
5. Select the sort mode of the result table.
6. By default, the results are listed in descending order, i.e. the largest value at the top. To change the sorting direction, enable "Sort ascending".
7. Define a maximum number of results to be displayed in the result table in the "Max. result count" field.

To display search zoom windows

1. In the "Search > Display" tab, enable "Show result zoom".

   This stops a running search and a running acquisition.
   The zoom area is indicated in the diagram that displays the source waveform of the search. The zoom window is displayed for the first result that was found.
2. If you need to adjust the search zoom area, you can drag the area or their edges on the screen. You can also enter the limits of the search zoom window in the "Search > Display > Define result zoom".
   Be aware, that the zoom window size is valid for all results of a search definition. If you change the settings drastically for one result, they may not be correct for the next search result you switch to.

   See also:
   - Chapter 7.1.3, "Zooming for details", on page 247

12 Data and file management

This chapter describes how to manage instrument settings, and measurement results like waveform data, numeric results and screenshots.

The "Save/Recall" dialog provides functions for saving and restoring data on the instrument. A naming pattern is available and can be adjusted to simplify a clear data storage.

The effect of the [Camera] key can be configured to save screenshots or reports.

12.1 Save and recall user settings

To repeat measurements at different times or perform similar measurements with different test data, it is useful to save the used instrument settings and load them again later. Furthermore, it can be helpful to refer to the instrument settings of a particular measurement when analyzing the results. Therefore, you can easily save the instrument settings of a measurement. In addition to the measurement-related settings, user-specific display settings and active reference waveforms can also be saved and loaded.

Access: "Menu" key > "Save/Recall" key > "Save" tab > "User settings"

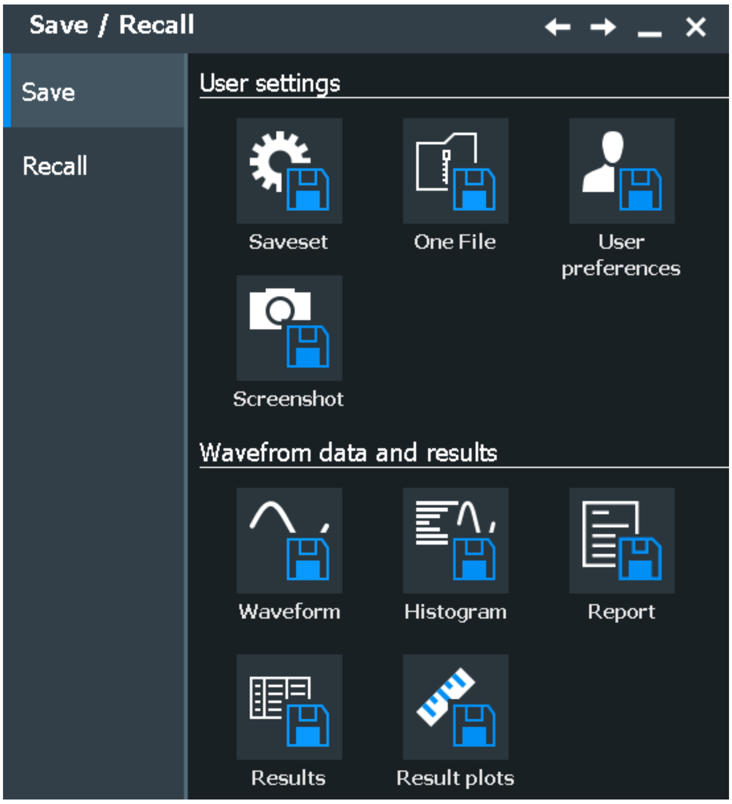


Figure 12-1: Save tab

Access to load instrument settings: "Menu" key > "Save/Recall" key > "Recall" tab > "User settings"

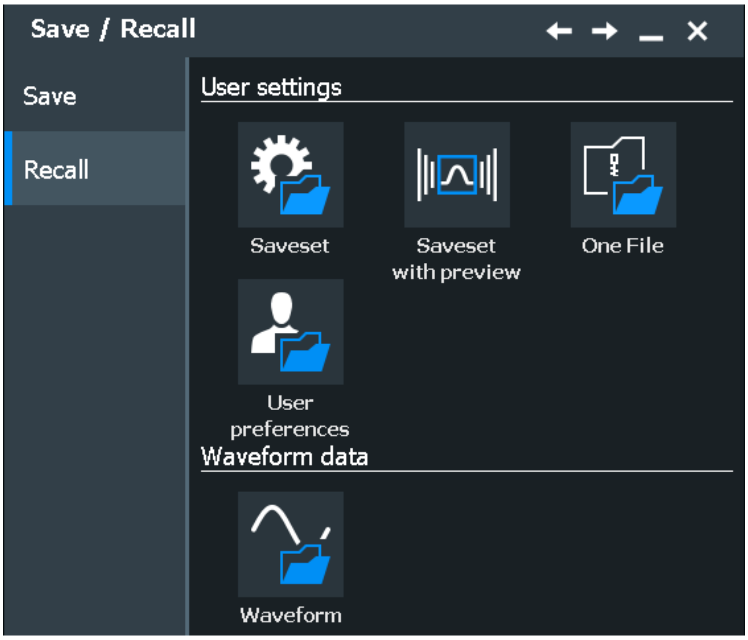


Figure 12-2: Recall tab

The R&S RTO provides the following types of saving and restoring settings:

- **Savesets** contain the complete instrument and measurement configuration except for user-specific display settings.
- **One File** contains the saveset and active reference waveform files in a ZIP file.
- **User preferences** contain the user specific display settings.
- **User-defined presets** contain the complete instrument setup including display settings, except for transparency and intensity. These settings can be restored by pressing the [Preset] key. See Chapter 4.6, "Preset setup", on page 103.

12.1.1 Savesets

Savesets contain the complete instrument and measurement configuration including a screenshot of the current display, but except for user-specific display settings stored as user preferences. You can save an unlimited number of setting files.

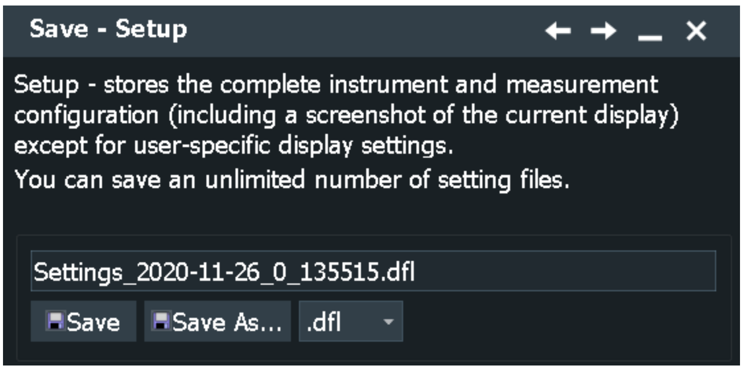
The waveform generator/pattern generator and the pulse source state is saved as "Off".

To load a saveset quickly, you can use the "Load saveset" function on the toolbar. A graphical preview helps you to find the required settings file.

If you need to store the instrument settings often, you can add the " Save settings" icon to the toolbar and use the icon to store the saveset file.

12.1.1.1 Save saveset settings

Access: "Menu" key > "Save/Recall" key > "Save" tab > "Savesets"



Save to file

Enter the file name to save the setting data to, and select the file format with the format button on the right. Double-tap the file name to open the file selection dialog box. See also: Chapter 12.5, "File selection dialog", on page 478.

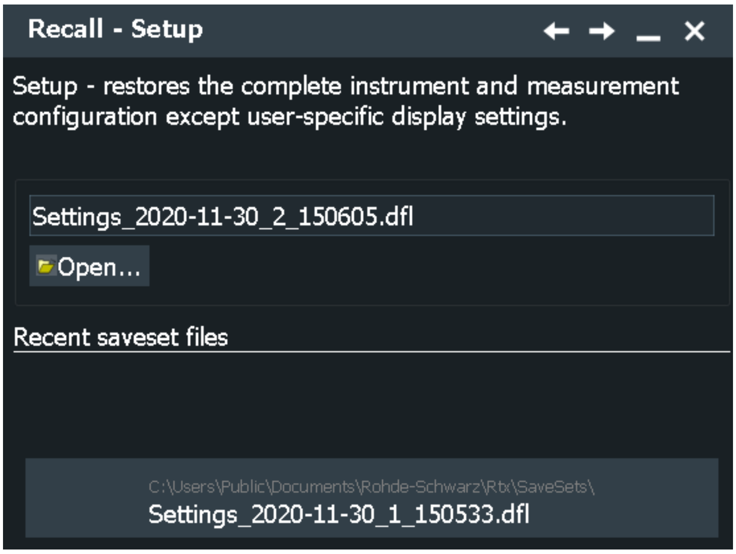
By default, saveset file names have the prefix "Settings_".

"Save"	Saves the data to the selected file.
"Save As..."	Opens the file selection dialog box and saves the data to the selected file.
".dfl/.xml"	Selects the file format.

Remote command:
`MMEMory:SAV` on page 1644

12.1.1.2 Load saveset settings

Access: "Menu" key > "Save/Recall" key > "Recall" tab > "Savesets"



Load from file

Enter the file name to load the setting data from, and select the file format with the format button on the right. Double-tap the file name to open the file selection dialog box. See also: Chapter 12.5, "File selection dialog", on page 478.

By default, saveset file names have the prefix "Settings_".

"Open"	Opens a file selection dialog box and loads the selected file.

Remote command:
`MMEMory:RCL` on page 1644

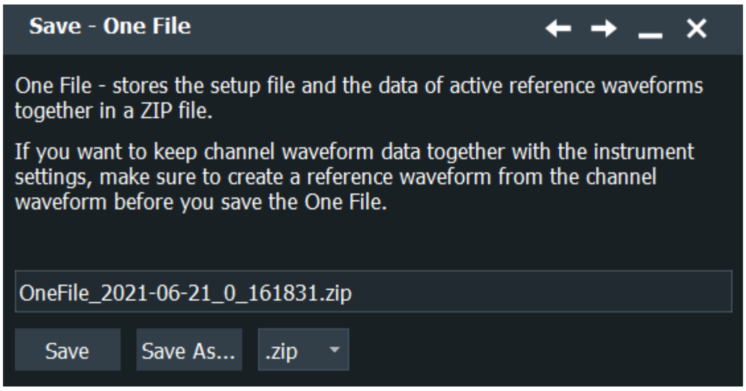
12.1.2 One File

The One File function stores the savesets file and the data of active reference waveforms together in a ZIP file. If you want to keep channel waveform data together with the instrument settings, make sure to create a reference waveform from the channel waveform before you save the One File.

The waveform generator/pattern generator and the pulse source state is saved as "Off".

12.1.2.1 Save One File settings

Access: "Menu" key > "Save/Recall" key > "Save" tab > "One File"



Save to file

Enter the file name to save the One File data to, and select the file format with the format button on the right. Double-tap the file name to open the file selection dialog box.
See also: Chapter 12.5, "File selection dialog", on page 478.

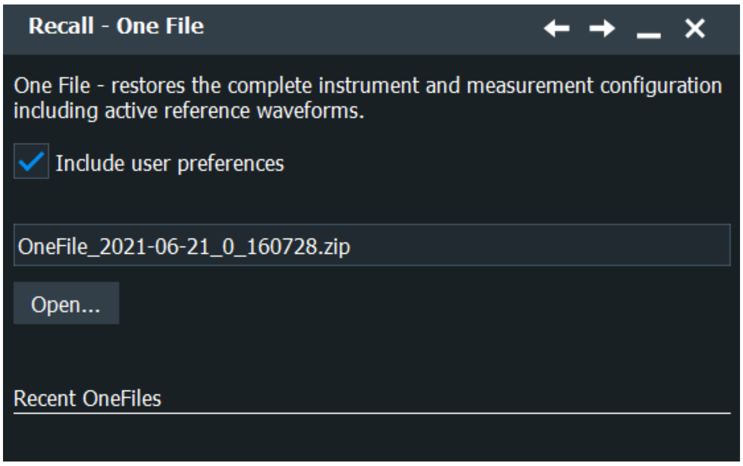
By default, one file names have the prefix "OneFile_".

"Save"	Saves the data to the selected file.
"Save As..."	Opens the file selection dialog box and saves the data to the selected file.
".zip"	Shows the file format.

Remote command:
`SAVeset:ONEFile:NAME` on page 1646
`SAVeset:ONEFile:SAVE` on page 1646

12.1.2.2 Load One File settings

Access: "Menu" key > "Save/Recall" key > "Recall" tab > "One File"



Load from file
Select "Open" to open the file selection dialog box, and to select the required file. See also: Chapter 12.5, "File selection dialog", on page 478.

By default, one file names have the prefix "OneFile_".

Remote command:
`SAVeset:ONEFile:NAME` on page 1646
`SAVeset:ONEFile:OPEN` on page 1646

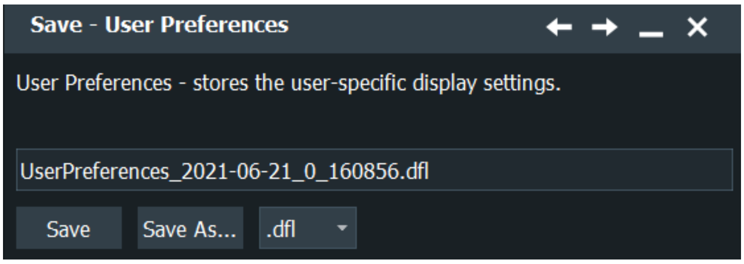
Include user preferences
If enabled, the user preferences settings are included in the one file. They include the user-specific display settings like diagram layout, toolbar, intensity and transparency settings.

12.1.3 User preferences

User preferences contain user-specific display settings like diagram layout, toolbar, intensity and transparency settings.

12.1.3.1 Save user preferences settings

Access: "Menu" key > "Save/Recall" key > "Save" tab > "User Preferences"



Save to file
The file name to save the data to.

By default, user preference file names have the prefix "UserPreferences_".

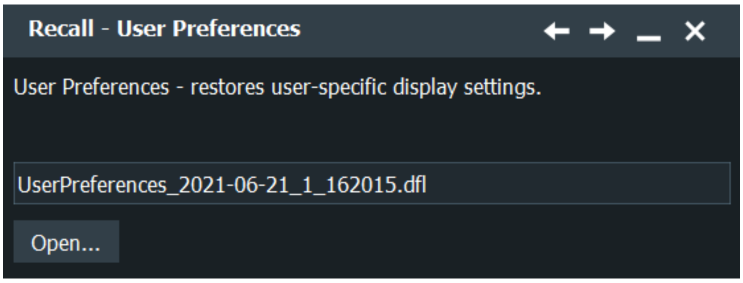
"Save" Saves the data to the selected file.

"Save As..." Opens the file selection dialog box and saves the data to the selected file.

".zip" Shows the file format.

12.1.3.2 Load user preference settings

Access: "Menu" key > "Save/Recall" key > "Recall" tab > "User Preferences"



Load from file
Enter the file name to load the setting data from, and select the file format with the format button on the right. Double-tap the file name to open the file selection dialog box.
See also: Chapter 12.5, "File selection dialog", on page 478.

By default, user preference file names have the prefix "UserPreferences_".

"Open" Opens a file selection dialog box and loads the selected file.

Remote command:
`MMEMory:RCL` on page 1644

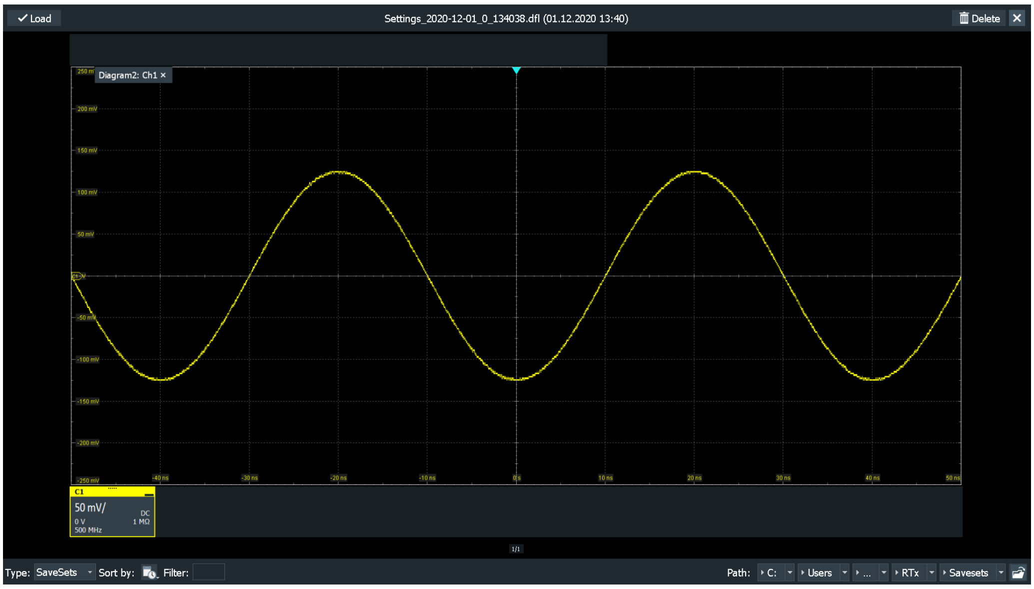
12.1.4 Saveset with preview function

Access: "Menu" > "Save/Recall" > "Recall" tab > "Saveset with preview"

The graphical preview helps you to find the required settings file or a reference waveform.

The "Saveset with preview" function is also available on the toolbar, see Chapter 3.3.6.3, "Toolbar functions", on page 57.

You can also assign the "Saveset with preview" function to the Quick Action key, see Chapter 4.5.1, "Hardkeys: function assignment", on page 99.



1. On the toolbar, tap the "Saveset with preview " icon.

   A window opens and shows the screenshot of the first measurement configuration that is stored in the default directory.

2. Select if you want to display a "SaveSets" file or a reference waveform ("Ref Wfm").

3. Find the required file using one of the following methods:
   - Tap the "Next" icon on the right or the "Previous" icon on the left to scroll the savesets of the directory.
     The file name is shown on the top, and the screenshot helps to identify the settings.
   - If the saveset was stored in another directory, use the path buttons at the bottom, or tap "Open" to open the required directory.
   - Tap "Sort by" to sort the files according to the "Last modified" date or the "File-name".
   - Specify a "Filter" name to show only the files containing the filter string.

4. Tap "Load" in the upper left corner to recall the settings of the selected file.

12.1.5 Saving and loading settings

Settings can be stored in a file with user-defined name and location, or in a quick saveset. The settings in a saveset can be saved and retrieved quickly at the touch of a button, so savesets are ideal for frequently used measurements.

For details on save/recall instrument settings and associated remote commands, see Chapter 12.1.1, "Savesets", on page 442.

To save settings to a saveset file

Alternatively, you can add the " Save settings" icon to the toolbar and use the icon to store the saveset file to the folder and file specified in the "Settings" tab.

See also Chapter 3.3.6.2, "Configuring the toolbar", on page 57.

1. Tap "Menu" and select "Save/Recall".
2. In the "Save" tab, press the "Saveset" button .
3. Tap "Save" to save the settings to the specified file.
   Tap "Save As" to save the settings to a different file. Select the file and directory from the file selection dialog box.

   The current settings are saved to the selected file.

To load settings from a saveset file

Alternatively, you can use the "Graphical Saveset" function on the toolbar.

1. Tap "Menu" and select "Save/Recall".
2. Select the "Recall" tab.
3. Press the "Saveset" button.
4. Tap "Load" to load the settings from the specified file.
   Tap "Open" to navigate to a different file. Select the file from the file selection dialog box and tap "Select".

   The saved settings are loaded to the R&S RTO.

To save a One File

1. If you want to save reference waveforms in the One File, create and display the reference waveforms.
2. Tap "Menu" and select "Save/Recall".
3. In the "Save" tab, tap the "One File".
4. Tap "Save" to save the settings to the specified file.
   Tap "Save As" to save the settings to a different file. Select the file and directory from the file selection dialog box.

   The current settings and the active reference waveforms are saved to the selected file.

To load a One File

1. Tap "Menu" and select "Save/Recall".
2. Press the "One File" button.
3. Tap "Load" to load the settings from the specified file.

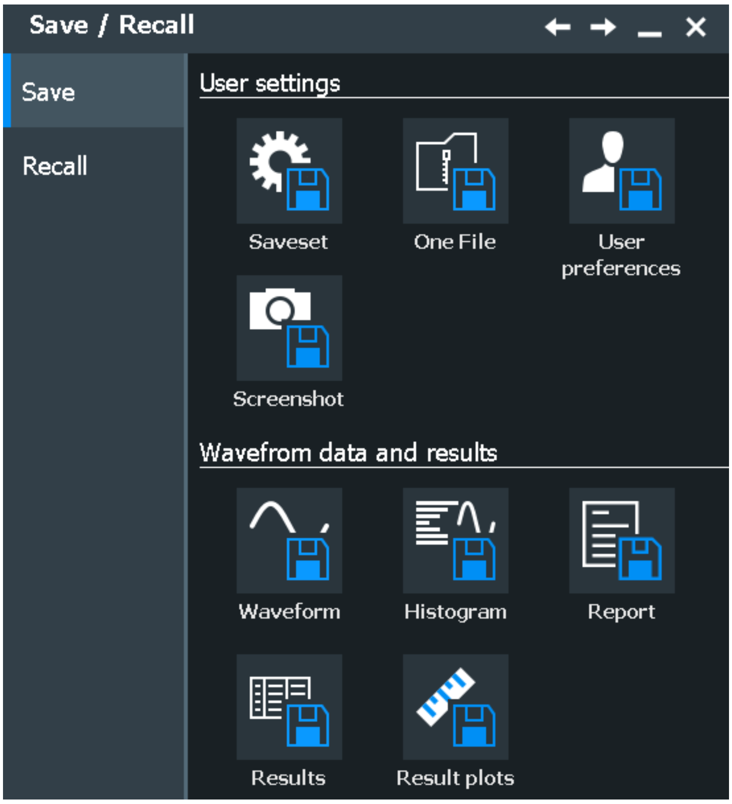
Tap "Open" to navigate to a different file. Select the file from the file selection dialog box and tap "Select".

The saved settings and reference waveforms are loaded to the R&S RTO.

12.2 Save and recall waveform data and results

Access: "Menu" key >"Save/Recall" key > "Save" tab

You can export various data to file: waveform data, histograms, and measurement results.



For information on data export in I/Q mode (requires option), see Chapter 17.3, "I/Q data output", on page 1061.

12.2.1 Waveform export files

Waveforms can be stored in XML, CSV, or BIN format.

Reloading waveforms: Restrictions

To reload waveform data as a reference waveform, it must be stored in BIN format.

If multiple acquisitions of one waveform are exported (Data logging or Multiple waveforms), only the first acquisition can be reloaded.

If the signal is a spectrum, reloading is only possible for waveforms with "Magnitude unit" = Linear. Waveforms with logarithmic unit cannot be reloaded.

Data of all waveforms is saved in two files. One file contains the waveform data values and is indicated by `*Wfm.*` in the filename. The second file contains the header data, for example, time scale, vertical scale, vertical and horizontal positions, interpolation mode and much more. Header data is required to restore the waveform from data, or to analyze the data values of the data file.

12.2.1.1 Header files

The header files of XML and BIN waveform files are written in XML format. The header files of CSV waveform files are written in CSV format. You can open the header files and use their information for data analysis.

CSV header files only contain the property names and values, one property per row.

```
Resolution:1e-010:
RecordLength:1000:
```

XML header files contain more information than CSV header files. The additional information is required to reload the stored waveforms with their correct settings.

```
<Prop Avail="0" ValueKey="" Name="Resolution" Value="1e-010" UserValue="0"
Step="1e-011" Default="0" Min="0" Max="1e+026" StepDefault="1e-011"
StepFactor="10" Resolution="0" UnitId="55" UnitName="s" UnitPowerProduct=""
BitGroupSize="0" Format="0"></Prop>

<Prop Avail="0" ValueKey="" Name="RecordLength" Value="1000" UserValue="1000"
Step="1" Default="1000" Min="0" Max="4294967295" StepDefault="1" StepFactor="10"
Resolution="1" UnitId="93" UnitName="Sa" UnitPowerProduct="" BitGroupSize="0"
Format="0"></Prop>
```

Header files contain the following properties:

Table 12-1: Header file properties

Value	Description
General	
FirmwareVersion	Firmware version that is installed on the R&S RTO (last entry in the header file)
Source	Name of the exported waveform

Value	Description
Resolution	Time between two samples *Resolution = 1 / Sample Rate*
SignalResolution	Time between two samples in this waveform. The value can differ from Resolution if the source is, for example, a spectrum, a bus signal, a correlation or a measurement. The value is determined automatically considering the waveform parameters and their dependencies. If the signal is a spectrum, the value indicates the frequency range of FFT bins.
EnhancementMode	Method to increase the sample rate if the required sample rate is higher than the ADC sample rate.
InterpolationMode	Interpolation method. The value is relevant when the enhancement mode is interpolated time.
DecimationMode	Method to reduce the number of data samples to achieve the required sample rate
DecimationFactor	Factor to the number of data samples to achieve the required sample rate *Decimation factor = ADC sample rate / Sample rate*
TraceArithmetics	Off, Envelope, or Average
InterleavedTraceCount	Number of y-values saved at each sampling time. The value is usually 1. The value is 2, if min and max values are saved for each sample, for example, for envelope waveforms.
SignalFormat	Format of the data values: ● FLOAT: floating point numbers, general export format ● INT (8 Bit): Integer 8 bit, used for "Raw (ADC sample)" data export. ● INT (16 Bit): Integer 16 bit, used for "Raw (ADC sample)" data export in high definition mode.
Timestamp	Absolute time of the waveform recording
ByteOrder	Endianness, only relevant for raw data export in high definition mode (SignalFormat = INT (16 Bit)). ● LSB first: little endian, least significant byte first ● MSB first: big endian, most significant byte first
NumericFormat	Number format of bus values and digital channel data (bit pattern format)
Record length	
RecordLength	Number of samples in a waveform record of one acquisition
HWRecordLength	Equivalent to the RecordLength
SignalRecordLength	Number of required samples in the waveform. The value can differ from RecordLength and HWRecordLength if the source is, for example, a spectrum, a bus signal, a correlation or a measurement. The value is determined automatically considering the waveform parameters and their dependencies. If the signal is a spectrum, the value indicates the number of FFT bins.
SignalHardwareRecordLength	Number of samples actually available in this waveform, including the number of required samples in the waveform and the additional samples needed for further computation

Value	Description
LeadingSettlingSamples	Relevant only for BIN files. In XML and CSV files, the value is 0. Number of additional samples before the beginning of waveform samples. These additional samples are needed for further computation, for example, for filters.
Horizontal system	
TimeScale	Horizontal scale in seconds per division
HorizontalDivisionCount	Number of horizontal divisions
RescaleCenterTime	Horizontal position, the time distance between the reference point and the zero point of the diagram
RescaleCenterPoint	Position of the reference point in % of the screen
ReferencePoint	Position of the zero point in % of the screen
TriggerOffset	Time distance from the trigger point to the zero point of the diagram
XStart	Horizontal start value of the waveform (time or frequency) *)
XStop	Horizontal stop value of the waveform (time or frequency)
HardwareXStart	Actual horizontal start value of data, including the settling time for further computation *)
HardwareXStop	Actual horizontal stop value of data, including the settling time for further computation
	*) If the waveform is a spectrum, the XStart and HardwareXStart values may be slightly smaller than the specified start frequency, or even get negative. The spectrum is centered on the center frequency, and the frequency range covered by one spectral bin is given by the SignalResolution. Hence, the spectral bin in the center of the spectrum always covers the range [CenterFrequency; CenterFrequency + SignalResolution[. As a result, the range covered by the first spectral bin in the spectrum may reach further than the specified start frequency. It is ensured that the specified start frequency is included in the frequency range.
Vertical system If multiple channels are exported, the values of channel1 are delivered, no matter if channel 1 is exported or not.	
VerticalScale	Vertical scale of the waveform in Volts per division, or other unit / division
VerticalDivisionCount	Number of vertical divisions
VerticalPosition	Vertical position of the waveform in divisions
VerticalOffset	Vertical offset of the waveform in Volts, or other unit
NofQuantisationLevels	Theoretical number of quantization levels in the signal. This value depends on the waveform format (8 bit, 16 bit, …). In case of a math waveform, it depends on the quantization levels of the operands and on the operator type.
BaseYStart	Vertical start value of the waveform
BaseYStop	Vertical stop value of the waveform

Value	Description
Multi channel export The header files contain strings like this: `MultiChannelVerticalOffset: 4:1.63:1.96:0:0:1e-005:0:-1e+026:1e+026:1e-005:10:0:V:`. Only the first 5 values and the unit at the end of the string are relevant for data analysis. All other values are for internal use and not explained here. Examples are in csv format.	
MultiChannelExport	Indication whether multiple channels are exported simultaneously: On | Off
MultiChannelExportState	Number of channels and export status of the individual channels, for example, 4:On:Off:On:On...: channels 1, 3 and 4 are exported.
MultiChannelVerticalOffset	Number of channels and vertical offset of the individual channels, for example, 4:0:0:0.02:0...: channel 3 has an offset of 20 mV.
MultiChannelVerticalPosition	Number of channels and vertical position of the individual channels, for example, 4:0:0:0:2...: the position of channel 4 is 2 divisions.
MultiChannelVerticalScale	Number of channels and vertical scale of the individual channels, for example, 4:0.05:0:0.03:0.04...: scale of channel is 50 mV/div, channel 3 has 30 mV/div and channel 4 has 40 mV/div.
MultiChannelBaseYStart	Number of channels and minimum value of the vertical range for each individual channel, for example, 4:-0.25:0:-0.13:-0.28...
MultiChannelBaseYStop	Number of channels and maximum value of the vertical range for each individual channel, for example, 4:0.25:0:0.17:0.12...: The range of channel 1 is -250 mV to 250 mV. The range of channel 3 is -130 mV to 170 mV. The range of channel 4 is -280 mV to 120 mV.
History	
TimestampState	State of the timestamps export. If on, the timestamps of each history waveform are written to the waveform data file.
Math waveform	
BaseUnit	Base unit of a mathematic waveform, for example, linear unit
ViewUnit	User-selected unit of a mathematic waveform, for example, logarithmic unit for a spectrum. The value is only valid if the exported waveform is a math waveform.
ViewUnitRelative	Indication of a relative unit. It is true if the math waveform has the ViewUnit "dB", for example. The value is only valid if the exported waveform is a math waveform.
ViewReferenceLevel	Reference level for a relative unit. The value is only valid if the exported waveform is a math waveform, and the unit is relative.
FFT	
CenterFreq	Center frequency of the spectrum
FreqSpan	Frequency span of the spectrum
FrequencyStart	Start frequency of the spectrum
FrequencyStop	Stop frequency of the spectrum
WindowType	Window used for the spectrum computation

Value	Description
ResolutionBW	Resolution bandwidth of the spectrum
AdjustedResolutionBW	Actual resolution bandwidth of a spectrum waveform. The value is only valid if the exported waveform is a spectrum.
GateRBWCoupling	Indication whether the record length or the resolution bandwidth is a constant for the spectrum computation
Parameters for power calculation	
Impedance	Impedance used for power calculation
NoiseBandwidth	Noise bandwidth of a spectrum waveform, required for power calculation. The value is only valid if the exported waveform is a spectrum.
Parameters for internal use	
SourceType	Source qualifier
TraceType	Waveform qualifier
ValueType	
TOADone	
BaseUnitRelative	Base unit indication
UseInterSampleTriggerOffset	
ISO_TRG SC_POST SC_TRG	

12.2.1.2 Waveform data files

The waveform data files - indicated by `*Wfm.*` in the filename - contain the actual waveform data. Usually only Y-values - mostly voltage values - are written subsequently. If the signal is a spectrum, the data of the last frame is written.

If the waveform consists of minimum and maximum values, two Y-values per sample are written, and the property `InterleavedTraceCount` in the header file is >1. This applies to envelope waveforms, for example.

The option "Interleaved X/Y" allows you to include horizontal values into the file.

If multi-channel export is enabled, the Y-values of the selected channels are written in interleaved order.

- One channel, single acquisition export
  - Normal waveform:
    Y_0; Y_1; Y_2; Y_3; ...
  - Envelope waveform:
    $Ymin_0$; $Ymax_0$; $Ymin_1$; $Ymax_1$; $Ymin_2$; $Ymax_2$; $Ymin_3$; $Ymax_3$; ...
  - Normal waveform, interleaved x/y data:
    X_0; Y_0; X_1; Y_1; X_2; Y_2; X_3; Y_3; ...
  - Envelope waveform, interleaved x/y data:

X_0;$Ymin_0$; $Ymax_0$; X_1; $Ymin_1$; $Ymax_1$; X_2; $Ymin_2$; $Ymax_2$; X_3; $Ymin_3$; $Ymax_3$; ...

- Multi-channel, single acquisition export
  In the example, two channels are exported.
  – Normal waveforms:
    $YCh1_0$; $YCh2_0$; $YCh1_1$; $YCh2_1$; $YCh1_2$; $YCh2_2$; $YCh1_3$; $YCh2_3$; ...
  – Envelope waveforms, channel 1 and channel 2 are envelopes:
    $YCh1min_0$; $YCh1max_0$; $YCh2min_0$; $YCh2max_0$; $Ymin_1$; $Ymax_1$; $YCh2min_1$; $YCh2max_1$; $Ymin_2$; $Ymax_2$; $YCh2min_2$; $YCh2max_2$; $Ymin_3$; $Ymax_3$; $YCh2min_3$; $YCh2max_3$; ...
  – Normal waveforms, interleaved x/y data:
    X_0; $YCh1_0$; $YCh2_0$; X_1; $YCh1_1$; $YCh2_1$; X_2; $YCh1_2$; $YCh2_2$; X_3; $YCh1_3$; $YCh2_3$; ...
  – Envelope waveform and normal waveform, interleaved x/y data:
    X_0;$YCh1min_0$; $YCh1max_0$; $YCh2_0$; X_1; $YCh1min_1$; $YCh1max_1$; $YCh2_1$; X_2; $YCh1min_2$; $YCh1max_2$; $YCh2_2$; X_3; $YCh1min_3$; $YCh1max_3$; $YCh2_3$; ...

In XML and CSV waveform value files, the data of each sample is grouped. The example shows the values of two samples for two waveforms and interleaved x/y data. The first waveform is an envelope, the second one is a normal waveform.

In CSV files, the data values for a given sampling time are written in one row.

```
-1.96e-008    -0.0079051387    -0.0059288535    -0.1027668
-1.95e-008    -0.0098814229    -0.0079051387    -0.10474309
```

In XML format, an empty line marks the beginning of the next sample.

```
<Data>-1.96e-008</Data>
<Data>-0.0079051387 </Data>
<Data>-0.0059288535 </Data>
<Data>-0.1027668 </Data>

<Data>-1.95e-008</Data>
<Data>-0.0098814229 </Data>
<Data>-0.0079051387 </Data>
<Data>-0.1027668 </Data>
```

If multiple acquisitions (Data logging / Multiple waveforms) are exported, the first acquisition is written in the same way as with single acquisition export. The following acquisitions are appended in the same way. If the signal is a spectrum, the last frame of each acquisition is saved.

If the history is exported in one file with timestamps, the timestamp as double float precedes the data of each acquisition. The acquisition data ist written in the same way as with single acquisition, according to the selected settings.

```
<timestamp as double float of first acquisition>
<value 1>
<value 2>
<value ...>
<timestamp as double float of second acquisition>
<value 1>
```

```
<value 2>
<value ...>
```

In BIN files, the instrument writes some leading and trailing settling samples before and after the waveform data. They ensure that the analysis of the reloaded reference waveform returns the same results as analysis of the original waveform. The number of leading settling samples is provided in the header file.

12.2.1.3 Number of samples in the waveform data file

In this section, a sample is defined as one or more values acquired at a given sampling time. The number of samples for one channel and acquisition is given in the header file by the property `SignalHardwareRecordLength`.

If the waveform has more than one Y-value per sample (e.g. envelope), the property `InterleavedTraceCount` is > 1, and the number of values in the file for this waveform is:

*No of values per waveform = InterleavedTraceCount * SignalHardwareRecordLength*

If multiple acquisitions are exported, the total number of values in the file is:

*No of values = InterleavedTraceCount * SignalHardwareRecordLength * No of exported acquisitions*

If "Interleaved x/y"is enabled, one horizontal value is added per sample. The total number of values in the file is:

*No of values = (1+ InterleavedTraceCount) * SignalHardwareRecordLength * No of exported acquisitions*

In BIN files, the value `SignalHardwareRecordLength` includes the number of required samples in the waveform and additional samples at the beginning (leading samples) and the end of the file (trailing samples). The number of additional samples is:

No of additional samples = SignalHardwareRecordLength - SignalRecordLength

The number of leading additional samples is given in the header file: `LeadingSettlingSamples`.

The number of trailing additional samples is:

No of trailing additional samples = No of additional samples - LeadingSettlingSamples
= SignalHardwareRecordLength - SignalRecordLength - LeadingSettlingSamples

MSO option R&S RTO-B1:

If the data of digital channels is stored in BIN format, 1 bit is written for each sample. 8 data samples are written in 1 byte (data word). Thus, the file size is

File size = Number of samples / 8

For example, 100 MSa are written into a 12.5 MByte BIN file. After reading the file, you have to extract the samples from the data words.

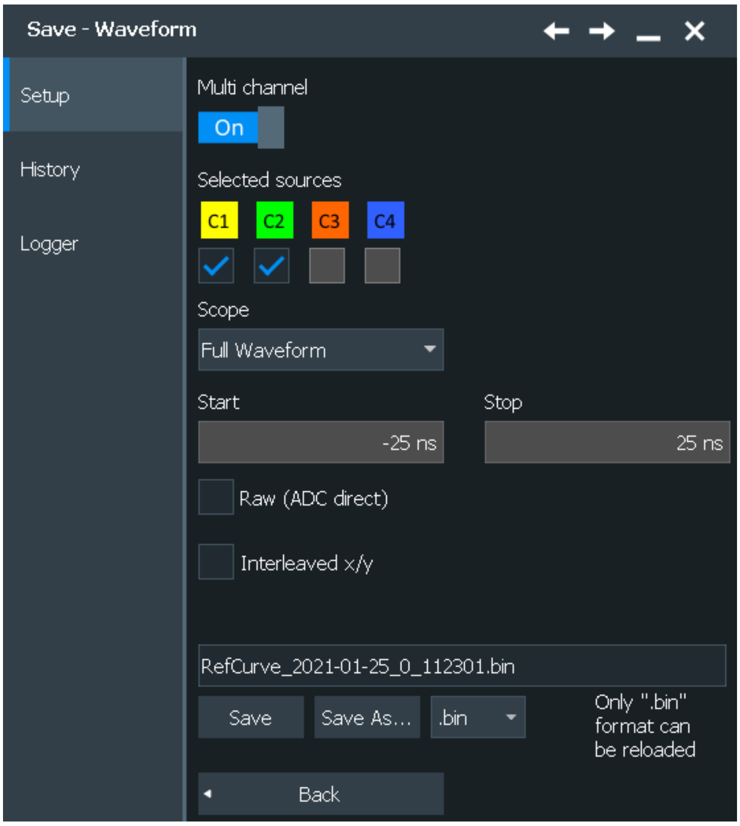
12.2.2 Waveform settings

In the "Save - Waveform" dialog, the storage settings for waveform data are defined.

See also: Chapter 12.2.6, "Saving and loading waveform data", on page 469.

12.2.2.1 Waveform setup settings

Access: "Menu" key >"Save/Recall" key > "Save" tab > "Waveform" > "Setup" tab.



Multi channel
Enables or disables the export of multiple input channels. If enabled, you can export the data of selected input channels (Selected sources) into one file.

If disabled, you can export one "Source" on page 457 waveform.

You can reload exported multiple channels if they are stored in BIN format. The import asks you to assign each stored waveform to a reference waveform.

Remote command:
`EXPort:WAVeform:MULTichannel` on page 1650

Source
Selects the waveform to be exported if "Multichannel export" is disabled.

Active waveforms of input channels, math signals, reference waveforms and advanced jitter analysis are available for export.

If the MSO option R&S RTO-B1 is installed, you can save also digital channels and parallel buses.

Remote command:
`EXPort:WAVeform:SOURce` on page 1648

Selected sources
Select the channels to be included in data export if "Multi channel" is enabled. Waveform1 of up to four input channels can be saved into one file.

Remote command:
`CHANnel<m>:EXPortstate` on page 1650

Scope
Defines the part of the waveform record that has to be stored.

"Full waveform"	Saves the complete waveform record.
"Zoom"	Saves the data included in the zoom area if at least one zoom is defined for the source waveform. The start and stop values of the area are shown. If several zooms are defined, select the "Zoom" to be used for export.
"Cursor"	Saves the data between the cursor lines if at least one cursor measurement is defined for the source waveform. The start and stop values of the area between the cursor lines are shown. If several cursor sets are defined, select the "Cursor set" to be used for export.
"Gate"	Saves the data included in the measurement gate if a gated measurement is defined for the source waveform. Select the "Measurement" for which the required gate is defined. The start and stop values of the gate are shown.
"Manual"	Saves the data between user-defined "Start" and "Stop" values.

Remote command:
`EXPort:WAVeform:SCOPe` on page 1651
`EXPort:WAVeform:STARt` on page 1652
`EXPort:WAVeform:STOP` on page 1652
`EXPort:WAVeform:ZOOM` on page 1652
`EXPort:WAVeform:CURSorset` on page 1653
`EXPort:WAVeform:MEAS` on page 1653

Raw (ADC direct)
Enables the export of analog channel data in the raw sample format of the ADC. The data format is integer 8 bit (signed 8-bit binary format). This format reduces the file size (1 Byte/sample instead of 4 Bytes/sample in binary files) but decreases the precision of the values.

If the high definition mode is active, the data format is integer 16 bit, except for peak detect decimation (8 bit). See "Export" on page 142.

Only y-values are exported, the "Interleaved x/y" option is not available.

Currently, the setting is not available for the export of digital channel data and data of R&S RT-ZVC channels.

Data conversion:

To convert INT8 or INT16 data to physical quantities, e.g. voltages, use the following formulas:

*ConversionFactor = VerticalScale * VerticalDivisionCount / NofQuantisationLevels*

*PhysicalQuantity = (Value_ADC * ConversionFactor) + VerticalOffset - (VerticalScale * VerticalPosition)*

The raw values are written in the `*.Wfm.*` file, all other values can be found in the corresponding header file.

Table 12-2: Example of raw data conversion

	INT8	INT16, HD mode
VerticalScale	110 mV/div	50 mV/div
VerticalPosition	1 div	0
VerticalOffset	50 mV	0
NofQuantisationLevels	253	253 * 256
VerticalDivisionCount	10	10
Value_ADC	13	-61
ConversionFactor	0.11 * 10 / 253 = 0.004347826086957	0.05 * 10 / (253 * 256) = 0.0000771986
Voltage	(13 * 0.004347826086957) + 0.05 - (1 * 0.11) = -0.003478260869559 V = 3.478260869559 mV	(-61 * 0.0000771986) + 0 = -4.7091146 mV

Remote command:
`EXPort:WAVeform:RAW` on page 1655

Interleaved x/y

Includes horizontal values in the export data (time or frequency values, depending on the waveform). X and Y-values are written alternately to the file. If disabled, only Y-values - mostly voltage values - are written.

Interleaved x/y data cannot be exported as raw values, the "Raw (ADC direct)" option is not available.

Remote command:
`EXPort:WAVeform:INCXvalues` on page 1654

Save to file

Enter the filename to save the waveform to. Double-tap the filename to open the file selection dialog box.

By default, the filename has the prefix "RefCurves_". You can define a pattern for automatic naming in the "Autonaming" dialog, see Chapter 4.8.1, "Autonaming", on page 110.

"Save" Saves the waveform as a reference waveform in the selected file.

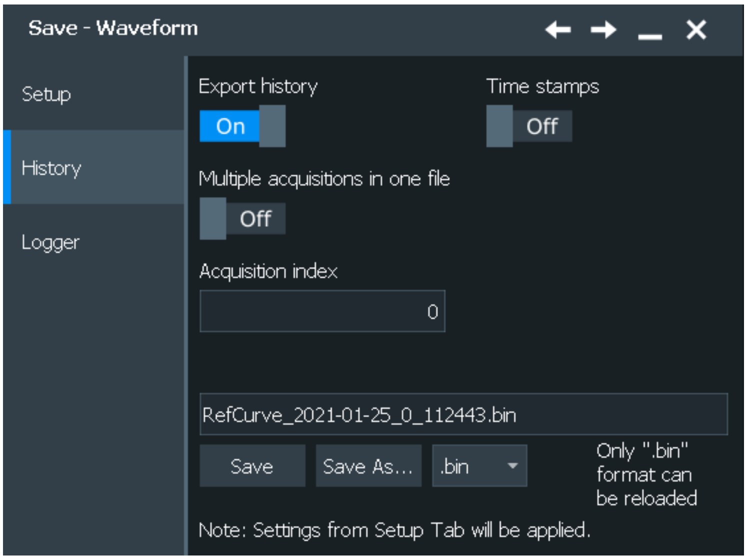
"Save As..." Opens the file selection dialog box and saves the waveform to the selected file. See also Chapter 12.5, "File selection dialog", on page 478

".bin/.xml/.csv" Selects the file format. Note that reference waveforms can be loaded from `.bin` files only.
See also: Chapter 12.2.1, "Waveform export files", on page 450.

Remote command:
`EXPort:WAVeform:NAME` on page 1651
`EXPort:WAVeform:SAVE` on page 1651

12.2.2.2 Waveform history settings

Access: "Menu" key >"Save/Recall" key > "Save" tab > "Waveform" > "History" tab.



In this dialog you can configure the export of the history waveform data to file. Select the channels and scope in the "Setup" tab.

Export history
Enables the history mode and the export of history waveforms to file.

Remote command:
`CHANnel<m>[:WAVeform<n>]:HISTory[:STATe]` on page 1463

Time stamps
If enabled, the relative timestamps of all history waveforms are written into the waveform data file at the beginning of each waveform record.

The following format is used:

```
<timestamp as double float of first acquisition>
<value 1>
<value 2>
<...>
<timestamp as double float of second acquisition>
<value 1>
<value 2>
<...>
```

You can also select the format of the time stamp in the "Acquire" > "History" dialog.

Remote command:
EXPort:WAVeform:TIMestamps on page 1654

Multiple acquisitions in one file
If enabled, allows you to save several or all history waveforms. Define the part of the history to be exported using "Start acquisition" and "Stop acquisition".

If multiple acquisitions of one waveform are exported into a BIN file, the first acquisition can be reloaded as reference waveform.

Remote command:
CHANnel<m>[:WAVeform<n>]:HISTory:STARt on page 1464
CHANnel<m>[:WAVeform<n>]:HISTory:STOP on page 1464

Start export
Starts the history replay and simultaneous saving.

Remote command:
CHANnel<m>[:WAVeform<n>]:HISTory:PLAY on page 1465

Save to file
Enter the filename to save the waveform to. Double-tap the filename to open the file selection dialog box.

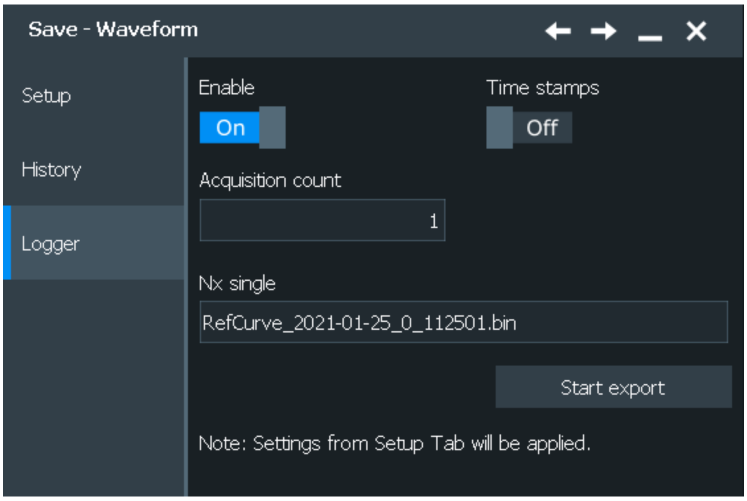
By default, the filename has the prefix "RefCurves_". You can define a pattern for automatic naming in the "Autonaming" dialog, see Chapter 4.8.1, "Autonaming", on page 110.

"Save"	Saves the waveform as a reference waveform in the selected file.
"Save As..."	Opens the file selection dialog box and saves the waveform to the selected file. See also Chapter 12.5, "File selection dialog", on page 478
".bin/.xml/.csv"	Selects the file format. Note that reference waveforms can be loaded from .bin files only. See also: Chapter 12.2.1, "Waveform export files", on page 450.

Remote command:
EXPort:WAVeform:NAME on page 1651
EXPort:WAVeform:SAVE on page 1651

12.2.2.3 Waveform logger settings

Access: "Menu" key >"Save/Recall" key > "Save" tab > "Waveform" > "Logger" tab.



In this dialog, you can configure the export of the data of a running Nx Single acquisition. Select the channels and scope in the "Setup" tab.

Enable

Enables the export of all waveforms of an Nx Single acquisition into one file.

The waveform records are written in historical order one after the other, either the complete records or the sections as defined in Scope. Set the number of acquisitions to be acquired and stored with "Acquisition count". The maximum amount of data that can be written is shown in "Max. file size".

Enabling "Data logging" stops a running acquisition. To start the logging, tap "Start Export" on page 462 or press [Single].

Pressing "Run cont" disables data logging.

Remote command:
`EXPort:WAVeform:DLOGging` on page 1653

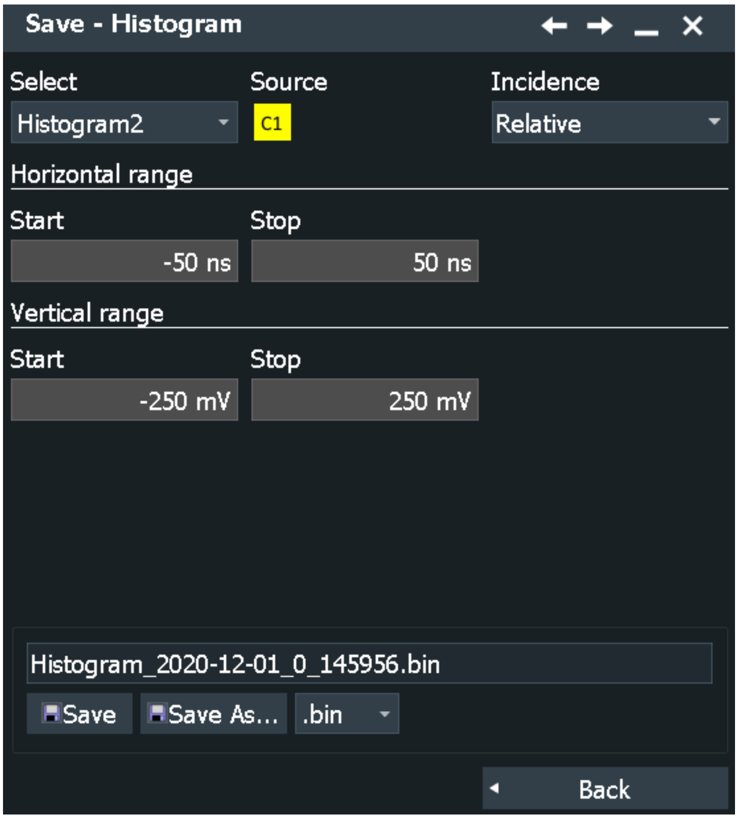
Start Export

Starts an Nx Single acquisition series and simultaneously saves the waveform data to a file.

Remote command:
`RUNSingle` on page 1326 (Nx Single acquisition)

12.2.3 Histogram

Access: "Menu" key >"Save/Recall" key > "Save" tab > "Histogram"



The waveform histogram export saves data in two files. The `*.Wfm.*` file contains 256 or 512 absolute or relative histogram values. The other file is the header file.

Contents of the header file:

- Source waveform of the histogram
- Histogram mode: vertical or horizontal
- Incidence of exported values: absolute or relative
- Histogram range: XStart, XStop, YStart, YStop
- Name of the exported histogram

Data conversion:

Using the header data, you can calculate the waveform value to which a histogram value belongs:

$$YValue = (YStop - YStart) / HistogramValuesCount * HistogramValueNumber + YStart$$

YStart	-0.25 V
YStop	0.25 V
HistogramValuesCount	256 (total number of written rows in a CSV file)
HistogramValueNumber	68 (number of the row in a CSV file)
Y-Value	(0.25 - (-0.25)) / 256 * 68 - 0.25 = -0.11719 V

The histograms settings are defined in the "Histogram" dialog box ("Menu" > "Apps" > "Analysis" tab > "Histogram").

See also: Chapter 8.2.8.4, "Histogram setup", on page 348.

Select
Selects the histogram to be exported. All active waveform histograms are shown in the list.

Measurement histograms can also be exported, see Chapter 12.2.5, "Result plots analysis", on page 466.

Remote command:
`EXPort:HISTogram:SELect` on page 1656

Source
Displays the source of the selected histogram.

Incidence
Sets the mode of exported histogram data: relative or absolute count of values. If relative values are exported, the sum of all values is 1, and the count of each value is set in relation to the sum.

Remote command:
`EXPort:HISTogram:INCidence` on page 1656

Horizontal range, Vertical range
Displays the "Start" and "Stop" limits of the histogram area.

Save to file
Enter the filename to save the waveform histogram to. Double-tap the filename to open the file selection dialog box.

By default, the filename has the prefix "Histogram_". You can define a pattern for automatic naming in the "Autonaming" tab. The default directory is:

`C:\Users\Public\Documents\Rohde-Schwarz\RTO\Histograms`

"Save"	Saves the histogram data in the selected file.
"Save As..."	Opens the file selection dialog box and saves the histogram data to the selected file. See also Chapter 12.5, "File selection dialog", on page 478
".bin/.xml/.csv"	Selects the file format.

Remote command:
`EXPort:HISTogram:NAME` on page 1656
`EXPort:HISTogram:SAVE` on page 1657
`EXPort:HISTogram:DATA?` on page 1657

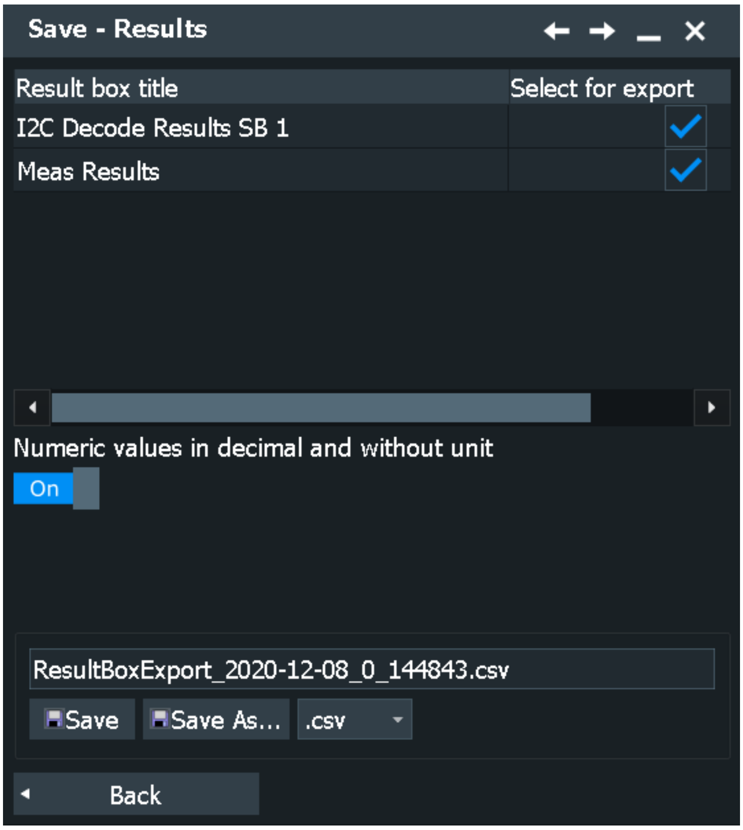
12.2.4 Results

In this tab, you can select the result boxes to be saved, and define the storage settings.

Access: "Menu" key >"Save/Recall" key > "Save" tab > "Results"

Access to the tab is available in all tabs where measurement and analysis settings are defined, for example, in the "Measurements Setup", "Cursors Setup", and "Masks Test Definition" tabs: Simply tap the "Export" button.

Furthermore, you can export the decode result tables of serial protocol analysis.



Result selection
The table lists all result boxes and decode tables that are currently open, including minimized boxes and docked boxes. Select the results that you want to save to file. All results are written into one file.

Note: If the result box is minimized, only the columns shown on the result icon are saved (2 columns). Statistical results are not shown on the minimized results icon, and they are not saved.

Remote command:
`EXPort:RESult:SELect` on page 1657

Numeric values in decimal and without unit
By default, numeric result values are written with their unit to the file. If the option is enabled, the values are saved with more decimal places.

Remote command:
`EXPort:RESult:NUMeric` on page 1658

Export results
Enter the filename to save the results to. Double-tap the filename to open the file selection dialog box.

By default, the filename has the prefix "ResultBoxExport_".

You can define a pattern for automatic naming in the "Menu" > "Settings" > "Save/Recall" > "Autonaming".

"Save" Saves the selected results to the indicated file.

"Save As..." Opens the file selection dialog box and saves the selected results to the selected file. See also Chapter 12.5, "File selection dialog", on page 478

".csv/.html" Selects the file format.

- CSV: the values are saved in a file
You can select the value delimiter and the list separator symbol in the "CSV Export" dialog, see Chapter 4.8.2, "CSV export", on page 112.
Tip for using MS Excel: It is recommended that you use the semicolon as csv decimal symbol. When you open the file with MS Excel, use "File > Open" and follow the wizard to set the separators correctly, or set the separator settings with "Tools > Options > International".
- HTML: Results are saved as web page for display in a browser.

Remote command:
`EXPort:RESult:NAME` on page 1658
`EXPort:RESult:SAVE` on page 1658

12.2.5 Result plots analysis

Access: "Menu" key >"Save/Recall" key > "Save" tab > "Result Plots"

You can export the data of long-term measurements, the measurement histogram and track data to file.

The measurement export saves results in two files. The `*.Wfm.*` file contains data values, and the other file is the header file.

The header file contains:

- Source waveform of the measurement
- Measurement scale
- Export type = Histogram, Long term or Track
- Exported measurement
- Histogram range: XStart, XStop, YStart, YStop
The range is only relevant for export type = histogram. The measurement axis is the X-axis, which can be a horizontal or vertical axis depending on the histogram mode.

Long-term measurements: The `*.Wfm.*` file contains one value or value set for each long-term measurement point. The maximum number of points is defined in the "Horizontal scaling" dialog box.

- If statistics are disabled, the current result of the main measurement is written - one double value per long-term point.
- If statistics are enabled, seven values for each long-term point are saved:
  – Upper peak
  – Lower peak
  – RMS
  – Standard deviation
  – Average
  – Event count per point: number of measurement results that creates one long-term point
  – Waveform count per point: number of waveforms included in one long-term point.

Measurement histogram: The `*.Wfm.*` file contains 1000 absolute or relative histogram values.

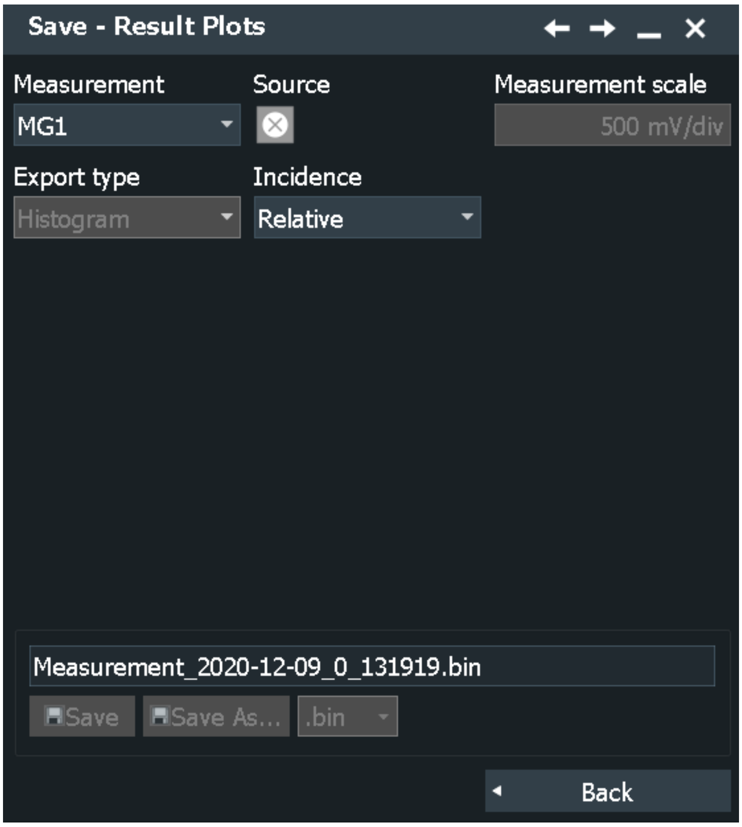
Data conversion of measurement histogram data:

Using the header data, you can calculate the measurement value to which a histogram value belongs:

*MeasValue = (XStop - XStart) / 1000 * HistogramValueNumber + XStart*

Example: The 273rd histogram value is 0.491749. That means, the relative frequency of the measurement value 0.1246 V is 0.491749.

XStart	0.07 V
XStop	0.27 V
HistogramValueNumber	273 (number of the row in a CSV file)
MeasValue	(0.27 - 0.07) / 1000 * 273 + 0.07 = 0.1246 V



Measurement

Selects the measurement to be exported.

Remote command:
`EXPort:MEASurement:SELect` on page 1658

Source, Measurement scale

Displays the measurement settings source and scale.

Export type

You can export the result data of the long-term measurement, the measurement histogram or the track.

To export the data, the required type must be enabled in the ""Menu" > "Measure" > Plot" tab: "Long term > Enable", "Histogram > Enable" or "Track > Enable".

Remote command:
`EXPort:MEASurement:TYPE` on page 1658

Incidence

Sets the mode of exported histogram data: relative or absolute count of values. If relative values are exported, the sum of all values is 1, and the count of each value is set in relation to the sum.

Remote command:
`EXPort:HISTogram:INCidence` on page 1656

Save to file

Enter the filename to save the measurement data to. Double-tap the filename to open the file selection dialog box.

By default, the filename has the prefix "Measurement_".

You can define a pattern for automatic naming in the "Autonaming" tab, see Chapter 4.8.1, "Autonaming", on page 110.

The default directory is:

`C:\Users\Public\Documents\Rohde-Schwarz\RTO\Measurements`

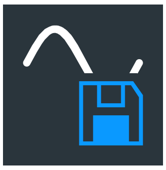
"Save"	Saves the measurement data in the selected file.
"Save As..."	Opens the file selection dialog box and saves the measurement data to the selected file. See also Chapter 12.5, "File selection dialog", on page 478
".bin/.xml/.csv"	Selects the file format.

Remote command:
`EXPort:HISTogram:NAME` on page 1656
`EXPort:MEASurement:SAVE` on page 1659
`EXPort:MEASurement:DATA?` on page 1659

12.2.6 Saving and loading waveform data

You can save the data of a channel, math or reference waveform to an `.xml`, `.csv`, or `.bin` file. The data export of several channels into one file is also possible. Files in `.bin` format can be reloaded to the R&S RTO as reference waveforms.

Instead of a complete waveform, you can also save a part of it, limited by a previously defined zoom, cursor lines, measurement gate or user-defined time values.



To save waveform data quickly, you can add the "Save Waveform" icon to the toolbar and use it for saving. The icon does not work for saving actions that are started with "Start export" (data logging and multiple history waveforms).

It is also possible to save history data to file. Furthermore, you can save a "live record" of a running RUN Nx SINGLE acquisition to one data file.

For details on waveform save/recall settings, see Chapter 12.2.2, "Waveform settings", on page 457.

The following procedures are described:

- "To save a waveform or a part of a waveform to a file" on page 469
- "To save a waveform using the toolbar icon" on page 470
- "To export waveform data of a running acquisition" on page 470
- "To save the history data" on page 283
- "To load a reference waveform" on page 253
  "To save a reference waveform" on page 252

To save a waveform or a part of a waveform to a file

1. Open "Menu" and select "Save/Recall" > "Save".

2. Tap the "Waveform" button.

   The "Save - Waveform" dialog opens.

3. Select the waveforms to be saved:
   - To save one waveform, tap the "Source" icon and select the waveform.
   - To save data of several channels, enable "Multi channel" and select the channels.

4. In the "Scope" list, select the part of the waveform record to be saved. Zoom, cursor and gate segments require the same setup for the selected waveform before saving. For "Manual", enter the "Start" and "Stop" time of the section.

5. Check the filename under "Save to file" and change it if needed. Usually, autonaming is used.

6. Check the file format and the "Export format" settings and change them if needed.

7. Tap "Save" to save the waveform data to the specified file.
   Tap "Save As" to save the waveform data to a different file or file type. Select the file from the file selection dialog box.

To save a waveform using the toolbar icon

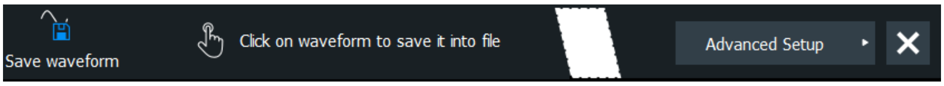
1. Add the "Save Waveform" icon to the toolbar, see Chapter 3.3.6.2, "Configuring the toolbar", on page 57.

2. Set the scope, export format, and other parameters in "Menu" > "Save/Recall" > "Waveforms".

3. If necessary, adjust the autonaming pattern and the storage path in "Menu" > "Settings" > "Save/Recall" > "Autonaming".

4. To save the waveform data:

   a) Tap the "Save Waveform" icon on the toolbar. The "Save Waveform" toolbar assist opens.

   b) Tap the waveform you want to save.
   If you tap the diagram background, the data of the focused waveform is saved.

To export waveform data of a running acquisition

1. Select the waveform you want to save and the scope as described in "To save a waveform or a part of a waveform to a file" on page 469, step 1 to 5.

2. If you want to save only a section of each waveform, set the "Scope".

3. Open the "Logger" tab.

4. Tap "Enable".

5. Enter the number of acquisitions to be acquired and saved in "Acquisition count".

6. Check the filename under "Nx single" and change it, if needed. Usually, autonaming is used.
7. Tap "Start export" to start the acquisition and to save the acquired waveform data to the specified file.

To load waveform data as a reference waveform

To reload waveform data from a previous measurement, the waveform must have been stored as a reference waveform in a BIN file before.

The procedure is described in Chapter 7.2.1, "Working with reference waveforms", on page 251.

12.3 Screenshots

To store the graphical results of the measurement, you can save a screenshot of the graphic area. To document current settings, the open dialog box can be included in the screenshot.

The "Image" toolbar icon saves the current display to a file according to the settings in "Menu" > "Save/Recall" > "Save" tab > "Screenshot". See also "Image" on page 59.

You can configure the [Camera] key to save screenshots by a single keypress. See also Chapter 4.5.1, "Hardkeys: function assignment", on page 99.

If a USB flash drive is connected to the instrument, the default path of the user data directory is set to the drive letter of the USB flash drive. Thus, you save data to USB flash drive automatically, and you can change the directory in the file explorer at any time.

Screenshots on a computer using the Web interface

If the R&S RTO is connected to a LAN, you can create and save screenshots of the instrument's display on a computer. See Chapter 22.3.2, "Web browser", on page 1231.

Meta information in screenshots

The meta data of the screenshot also contains instrument information. In PNG and JPEG files, meta information is saved as EXIF information and can be read, for example, using the ExifTool.

Example:

Reading meta information using the ExifTool.

Command: `# exif C:\Screenshot_2020-07-14_0_110551.png`

Result:

```
ExifTool Version Number         : 10.20
File Name                       : Screenshot_2016-07-14_0_110551.png
Directory                       : C:/
File Size                       : 37 kB
File Modification Date/Time     : 2020:07:14 11:05:51+02:00
File Access Date/Time           : 2020:07:14 11:05:51+02:00
File Creation Date/Time         : 2020:07:14 11:05:51+02:00
...
Instrument Firmware Version     : 3.30.0.46
Instrument Material Number      : 1329.7002k44
Instrument Serial Number        : 123456
Image Size                      : 1280x800
Megapixels                      : 1.0
```

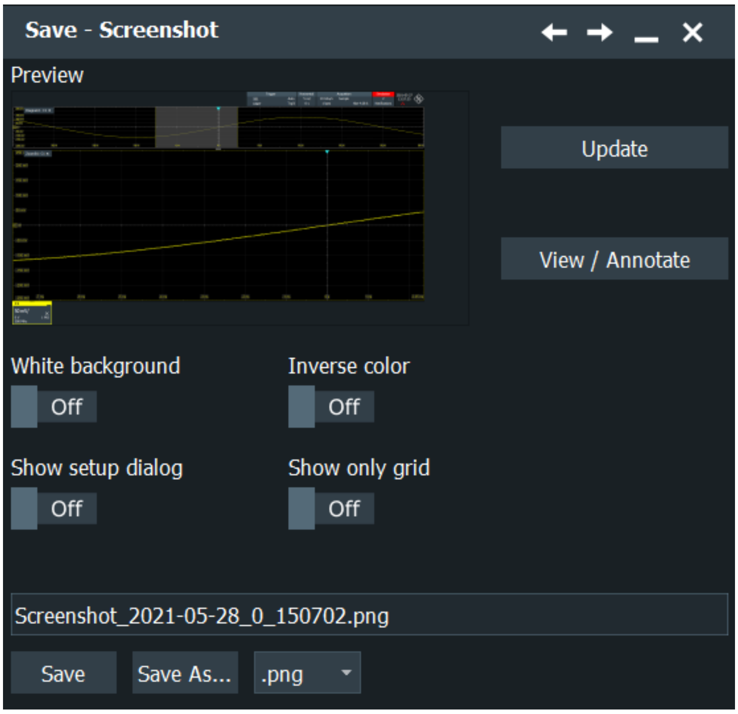
12.3.1 Screenshot settings

Access: "Menu" > "Save/Recall" > "Save" tab > "Screenshot"

In the "Screenshot" dialog box, you configure the image to be saved or included in a report. You select the storage location for screenshot files. The image is created when you open the dialog box, and can be updated at any time.

You can also edit the image before saving, and include an open dialog box or the side-bar in the image.

You can save the image in the "Screenshot" dialog box. To save screenshots quickly, use the "Image" toolbar icon, or configure and use the [Camera] key.



Preview

Shows a preview of the screenshot. The image is created when the dialog box opens.

Update

Updates the preview of the screenshot with the current display view, e.g. after changes to the settings have been made, or an additional channel has been activated.

White background

Inverts the background color. So you can picture waveforms with normal waveform colors on white background.

If both "White background" and Inverse color are enabled, the instrument inverts the background twice, and it appears black.

"White background"	"Inverse color"	Background	Waveform and results
On	Off	White	Screen colors
Off	On	White	Inverted colors

"White background"	"Inverse color"	Background	Waveform and results
On	On	Black	Inverted colors
Off	Off	Black	Screen colors

Remote command:
HCOPy:WBKG on page 1662

Inverse color
Inverts the colors of the output, i.e. a dark waveform is shown on a white background.

See also: "White background" on page 473.

Remote command:
HCOPy:DEVice<m>:INVerse on page 1661

Show setup dialog
If you want to save dialog boxes in screenshots, enable "Show setup dialog". The currently open dialog box is included in the screenshot.

Remote command:
HCOPy:SSD on page 1662

Show only grid
If enabled, the screenshot shows only the grid. The signal bar is not included.

Remote command:
HCOPy:ISBA on page 1663

View/ Annotate
Opens the screenshot in the Paint application. Edit the image as necessary. You can store the file using "Save as". Alternatively, save the file and close the Paint application to return to the "Screenshot" dialog, then save the edited image. The changes are not shown in the preview.

Save to file
Defines the filename to which screenshot is saved. By default, the filename has the prefix "Screenshot_". Double-tap the filename filed to change the name.

If a USB flash drive is connected to the instrument, the default path of the user data directory is set to the drive letter of the USB flash drive. Thus, you save data to USB flash drive automatically, and you can change the directory in the file explorer at any time.

"Save"	Saves the current screenshot to the specified file.
"Save As..."	Opens the file selection dialog box. Here you can adjust the target directory and the file name and save the current screenshot to the file. The symbols of important target folders are listed on the left of the file explorer.
"Delete"	Opens the file selection dialog box and deletes the selected file.

Remote command:
`HCOPy:DEVice<m>:LANGuage` on page 1661
`HCOPy:DESTination<1..2>` on page 1660
`MMEMory:NAME` on page 1661
`HCOPy:IMMediate<m>[:DUM]` on page 1663
`HCOPy:IMMediate<m>:NEXT` on page 1663
`MMEMory:DELete` on page 1642

12.3.2 Configuring and saving screenshots

You can edit the image, invert all colors, and set the background color. A preview of the current image is shown for reference.

1. Open the "Menu" > "Save/Recall" > "Save" tab > "Screenshot".
2. To enhance the images for later print on white paper, enable "White background" or "Inverse color". If you print this image later on a monochrome printer, you get a grayscaled picture. The contrast of the resulting gray lines depends on waveform colors and the used printer.
3. Select the file format: png, jpg, or another one.
4. To change the directory, tap "Save As" and configure the path.

   The symbols of often used target folders are listed on the left of the file explorer. By default, screenshots are saved in the `C:\Users\Public\Public Documents\Rohde-Schwarz\RTx\ScreenShots` directory.
5. Tap "Save".

   The file is saved and the dialog box closes.
6. Check if the screenshot is saved to the desired directory.
7. To save further screenshots, use one of the following ways:
   - Configure the [Camera] key. Press the key to save a screenshot.
     See also Chapter 4.5.1, "Hardkeys: function assignment", on page 99.
   - Add the "Image" icon to the toolbar. Tap the icon to save an image.
     See also Chapter 3.3.6.2, "Configuring the toolbar", on page 57.
   - Tap "Save" in the "Screenshot" dialog box to save the image to the specified file.
   - To save the image with a dedicated filename or to another directory, tap "Save As" in the "Screenshot" dialog box. Select the path, enter a filename, and tap "Save".

12.4 Reports

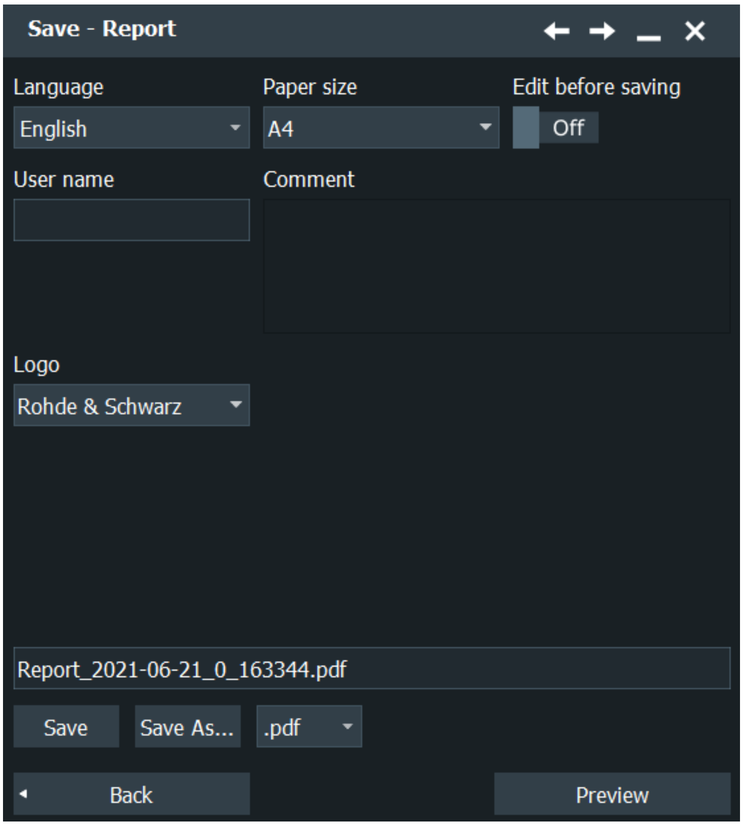
Reports document the current measurement and test results. The report contains general information, current vertical and horizontal settings, trigger settings, active channels and all current results except for zoom and search results. A screenshot is also included.

You can create the report manually, or automatically on defined events:

- Press the [Camera] key.
- Tap the "Create report" toolbar icon.
  Before, add the icon to the toolbar, see also Chapter 3.3.6.2, "Configuring the toolbar", on page 57.
- Action on micro button, available on active Rohde & Schwarz probes
- Action on trigger
- Event action at mask testing
- Event action at limit checks

12.4.1 Report settings

Access: "Menu" menu > "Save/Recall" menu > "Save" tab > "Report"



Language
Selects the language to be used in the report. Available languages are listed in the data sheet.

Remote command:
REPort:LANGuage on page 1664

Paper size
Selects the paper size: A4 or US Letter.

Remote command:
REPort:PAPersize on page 1664

Edit before saving
Enables you to edit the report info when you save reports using the [Camera] key. When you press the key, a dialog box opens where you can change the user name and the comment.

User name / Comment
Enter information that appears in the general information section at the beginning of the report.

Remote command:
REPort:USER on page 1665
REPort:COMMent on page 1665

Logo
By default, the Rohde & Schwarz logo is shown in the header of the report pages. You can switch the logo off, or select your logo to be shown. A preview of the selected log file is shown.

Remote command:
REPort:LOGType on page 1664
REPort:LOGO on page 1664

Save to file
Select the file format and define the filename of the report file. By default, the filename has the prefix "Report_".

Double-tap the filename to open the file selection dialog box.

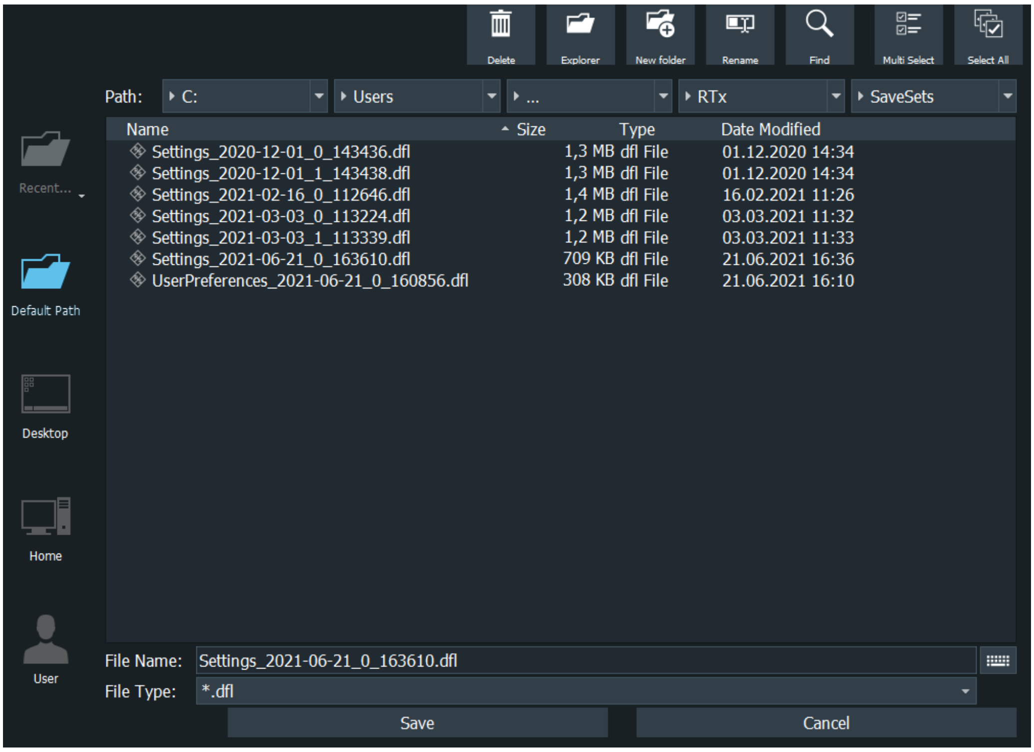
"pdf/doc/html"	Selects the report format.
"Save"	Saves the current report to the specified file.
"Save As..."	Opens the file selection dialog box and saves the report to the selected file.

Remote command:
REPort:FILE:NAME on page 1665
REPort:FILE:SAVE on page 1665

Preview
Opens the current report in PDF format.

12.5 File selection dialog

The file selection dialog provides a file explorer from which you can select a file to load or to save data to. You can also manage your files in this dialog.



Path
Tap the path elements to change the current folder. The default folder is defined in Default path for all file operations.

You can save the data in a local folder on the instrument, to an external storage device (usually a USB flash drive), or to a folder on a connected network drive. The path list provides all available drives and folders.

On the left, shortcut icons provide access to often used folders.

Toolbar
The toolbar on the top provides various functions for file management.

File Name
The file name to be loaded or stored to. Double-tap the file name, or tap the keyboard icon to enter the file name using the online keyboard.

The default file name for new files is defined in the "Autonaming" tab, see Chapter 4.8.1, "Autonaming", on page 110.

File Type
The file extension of the file to be loaded or stored to.

Save, Select
Selects the specified file for the open or save operation and closes the dialog box.

Cancel
Closes the dialog box without selecting a file.

13 Protocol analysis

Using the serial protocol options for the R&S RTO, you can analyze various serial protocols.

13.1 Basics of protocol analysis

Before you can analyze a serial signal, the bus has to be configured according to the protocol and specifics of the signal. The configuration contains:

- Assignment of the data and clock lines to the input channels
- Logical thresholds
- Protocol-specific settings

Serial data can be analyzed in several ways:

- Triggering: You can trigger on various events that are typical for the selected protocol type, for example, on start and stop of messages, or on specified data patterns in the message.

Triggering on a trigger event sequence is not supported, and holdoff settings are not available.
Triggering on SPI, I²C and UART protocols is included in the basic firmware, no option is required.

- Protocol decoding: The digitized signal data is displayed on the screen together with the decoded content of the messages in readable form, and the decode results are listed in a table.
- Search on decoded signal data: For most serial protocols, you can find various events in the decoded data. You can find the same events that you can trigger on, and even many more, because several event types can be combined. Thus, you get the results for the complete acquisition cycle.

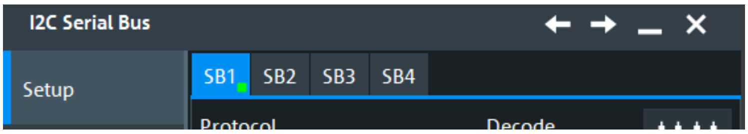
13.1.1 Setup - general settings

For all protocols, configuration starts with the selection of the serial bus and the protocol.

Table 13-1: Configuration settings are protocol-specific. They are described in the related chapters:

ARINC 429	Chapter 13.10.2, "ARINC 429 configuration", on page 619
Audio	Chapter 13.8.2, "Audio configuration", on page 585
CAN, CAN FD	Chapter 13.5.2, "CAN and CAN FD configuration", on page 525
Custom: Manchester / NRZ	Chapter 13.18.2, "Custom: Manchester / NRZ configuration", on page 783
CXPI	Chapter 13.26.2, "CXPI configuration", on page 942
Ethernet 10BASE-T and 100BASE-TX	Chapter 13.11.2, "Ethernet configuration", on page 631
Ethernet 100BASE-T1	Chapter 13.12.2, "100BASE-T1 configuration", on page 648
Ethernet 1000BASE-T1	Chapter 13.13.2, "1000BASE-T1 configuration", on page 666
FlexRay	Chapter 13.7.1.1, "FlexRay configuration", on page 566
I²C	Chapter 13.2.2, "I²C configuration", on page 492
LIN	Chapter 13.6.2, "LIN configuration", on page 552
MDIO	Chapter 13.20.2, "MDIO configuration", on page 826
MIL-1553	Chapter 13.9.2, "MIL-STD-1553 configuration", on page 603
MIPI D-PHY	Chapter 13.16.2, "D-PHY configuration", on page 737
MIPI M-PHY	Chapter 13.17, "M-PHY and USB SSIC (option R&S RTO-K44 and K64)", on page 752
MIPI RFFE	Chapter 13.15.2, "RFFE configuration", on page 717
SENT	Chapter 13.14.2.1, "SENT configuration", on page 687
SpaceWire	Chapter 13.24.2, "SpaceWire configuration", on page 900
SPI	Chapter 13.3.2, "SPI configuration", on page 505
UART	Chapter 13.4.2.1, "UART configuration settings", on page 516

USB	Chapter 13.21.2, "USB 2.0 configuration", on page 843
USB 3.1	Chapter 13.22.2, "USB 3.1 configuration", on page 874
USB PD	Chapter 13.23.2, "USBPD configuration", on page 888
USB SSIC	Chapter 13.17.2, "M-PHY configuration", on page 753
8b/10b	Chapter 13.19.2, "8b/10b configuration", on page 812
DDR3	Chapter 13.27.1, "DDR configuration", on page 960



Make sure that the tab of the correct serial bus is selected.

Protocol
Defines the protocol type of the selected serial bus.

Remote command:
`BUS<m>:TYPE` on page 1669

Decode
Enables the decoding of the selected bus. The signal icon of the bus appears on the signal bar.

Note: Exception for I²C, SPI, and UART signals:. The decode function is only available if the appropriate protocol option is installed. The bus can be used as trigger source without any option, as long as the bus is configured correctly.

Remote command:
`BUS<m>[:STATe]` on page 1669

13.1.2 Autoset for protocols

If you want to perform a quick protocol measurement, you can use the "Autoset" function. It allows you to find the correct instrument settings for the desired protocol. It displays at least one decoded frame after the execution.

Performing a protocol autoset

1. Press [Protocol] on the front panel.
2. Select the bus you want to configure.
3. Select the "Setup" tab.
4. Tap the "Protocol" you want to configure.
5. Assign the sources for the input channels.

6. Press the "Autoset" button.

"Autoset" performs the following steps:
- Executes "Autoset" for the horizontal and vertical scale.
- Adjusts the horizontal scale to display at least one frame or packet .
- Executes "Find thresholds" to determine the thresholds.
- If necessary, performs bit rate estimation.
- Sets default protocol trigger (frame start) .
- Turns on the protocol decoder.

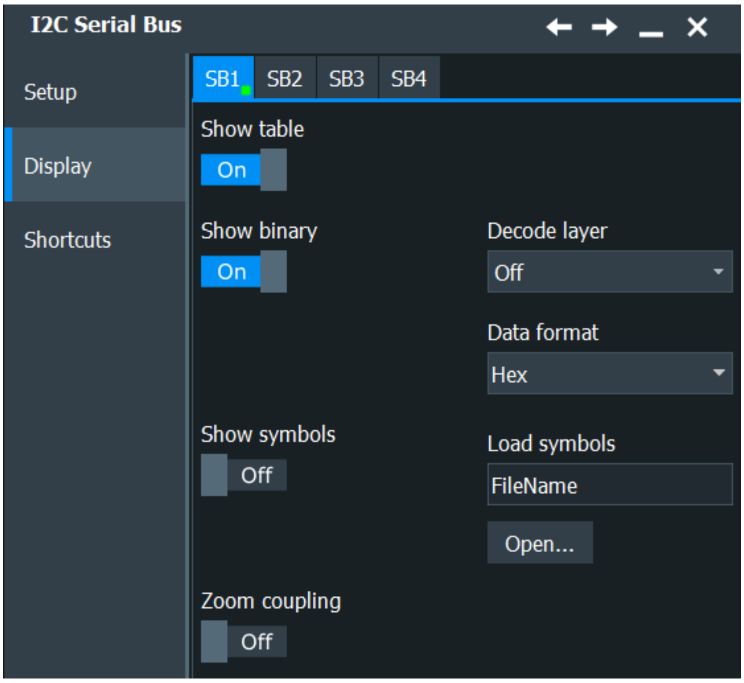
Remote command:

`BUS<m>:FAUToset` on page 1670

There are some limitations for the autoset:
- It is not available, if the digital MSO channels are selected as a source for the waveform.
- Signals with longer idle phases might not find horizontal adjustment.
- On duplex signals, different forward/reverse amplitudes can fail to determine thresholds.

13.1.3 Display

For all protocols, you can select to display the decoded signal as a table and to show the binary signal on the screen.



For some protocols, the result table provides a button to show the details of the selected frame.

Label Lists

For all protocols using ID or address identification, it is possible to create label lists containing addresses or IDs, a symbolic name for each node (symbolic label), and some protocol-specific information.

You can load label lists, and activate its usage for decoding. As a result, an additional "Label" column appears in the "Decode results" table, containing the symbolic label. The frame captions of the decoded signal show the symbolic label instead of the ID or address values. Hence it is easy to identify the messages of the different bus nodes.

You can also use the label list to trigger on an identifier or address. Instead of entering the value, you select the name, which is defined in the label list.

Show table
Opens a table with decoded data of the serial signal. The function requires the option for the analyzed protocol.

Decode results are protocol-specific. They are described in the "Decode Results" chapter of each protocol.

Remote command:
`BUS<m>:RESult` on page 1670

Show binary
For each configured line, the binary signal is also displayed.

Data format
Sets the data format for the values displayed in the decode table and in the combs of the decoded signal.

Remote command:
`BUS<m>:FORMat` on page 1671

Show symbols
Activates the symbols list to be used for decoding. The "Label" appear in the "Decode results" table and in the frame captions of the decoded signal.

Remote command:
`BUS<m>:SYMBols` on page 1671

Load symbols
Selects and loads a label list file. Available file formats are PTT, CSV, DBC (CAN only), and XML (FIBEX files, FlexRay only).

Label lists are protocol-specific. Their contents are described in the corresponding protocol chapters.

Remote command:
`BUS<m>:NEWList` on page 1671

Zoom coupling
If enabled, the decode zoom and result table are synchronized. If you select a row in the result table, this result is shown in the decode zoom.

With "Zoom window" you can also select the zoom used for coupling.

For an introduction to zoom settings and setup, see:
- Chapter 7.1.2, "Zoom settings", on page 243
- Chapter 7.1.3, "Zooming for details", on page 247

Remote command:
`BUS<m>:ZCOupling` on page 1671

13.1.4 Content and format of the PTT file

Label lists are stored as PTT (protocol translation table) files. The PTT file format is an extension of the CSV format (comma-separated values). You can edit it with standard editors, for example, with MS Excel or a text editor.

The PTT file has three types of lines:
- Comment lines begin with a hash character #. A hash character at any other position in the line is treated like a standard character.
- Command lines begin with a commercial at character @. An @ character at any other position in the line is treated like a standard character.
- Standard lines are the lines that not qualify as comment or command lines. They build the core of the label list.

Command lines

Command lines define the version of the PTT file and the protocol name:
- @FILE_VERSION: must appear exactly once in the file
- @PROTOCOL_NAME: must appear at least once in the file. Thus, one file can contain several label lists for different protocols.

```
# --- Start of PTT file
@FILE_VERSION  = 1.0
@PROTOCOL_NAME = i2c
[... Label list for I2C]
@PROTOCOL_NAME = can
[... Label list for CAN]
# --- End of PTT file
```

Standard lines

Standard lines define the contents of the label list. The rules for standard lines follow the csv convention, they are:
- Values are separated by commas
- Space characters following a delimiter are ignored
- Values with a special character (comma, newline, or double quote) must be enclosed in double quotes

- Text in double quotes must be escaped by double quote characters

The format of the numeric value is indicated by a suffix. The following formats are supported:

Format	Suffix	Example
Decimal	<empty> d	`106, DeviceName` `106d, DeviceName`
Hexadecimal	h	`6Ah, DeviceName` or prefix: `0x6A, DeviceName`
Octal	o	`152o, DeviceName`
Binary	b	`01101010b, DeviceName`

The maximum supported word size for (unsigned) integers is 64 bits.

```
# --- Start of PTT file
@FILE_VERSION  = 1.0
@PROTOCOL_NAME = i2c
#    Following two lines are equal:
7,01h,Temperature
7,01h, Temperature
#    A comma must be enclosed in double quotes:
7,01h,"Temperature, Pressure, and Volume"
#    A double quote must also be enclosed in double quotes:
7,7Fh,"Highspeed ""Master"" 01"
#    Following lines yield the same result:
7d,0x11,Pressure
7h,11h,Pressure
0x7,17d,Pressure
7,17,Pressure
```

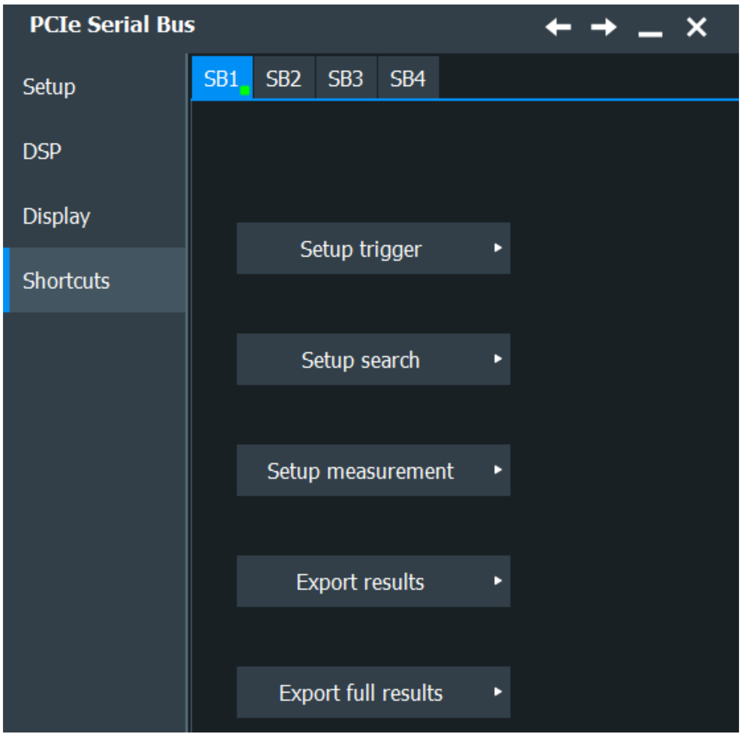
13.1.5 Shortcuts

The "Shortcuts" tab gives quick access to other dialogs with protocol relevant settings. The availability depends on the functions supported by the protocol.

If you access the dialog, from the "Shortcuts" tab, some settings are already predifined. For example, the trigger source is already set to the correct serial bus.

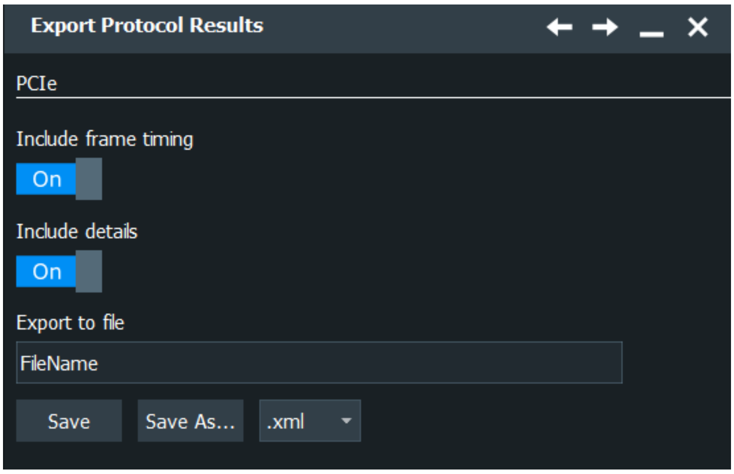
- "Trigger" dialog: you can trigger on various events that are typical for the selected protocol type, for example, on start and stop of messages, or on specified data patterns in the message.
- "Search" dialog: in this dialog you can configure prtocol specific search settings.
- "Measurement" dialog: in the "Measurements" dialog you can perform addiitonal measurements for some protocols. See Chapter 8.2.10, "Protocol measurements (option R&S RTO-K35)", on page 350.

- "Export reults": in the "Save Results" dialog, you can select the decode results you want to export, the file format, and the delimiter.
  For details, see Chapter 12.2.4, "Results", on page 464.
- "Export full results": in this dialog you can select the decode results you want to export and the protocol specific details to be included.



13.1.6 Export protocol results

In the "Export Protocol Results" dialog, you can export the results and all details of the selected protocol.



Include frame timing
Includes the frame timing.

Remote command:
`BUS<m>:EXPResult:TIME` on page 1672

Include details
Includes the detailed results for all frames.

Remote command:
`BUS<m>:EXPResult:DETail` on page 1672

Export to file
Enter the filename to save the results to. Double-tap the filename to open the file selection dialog box.

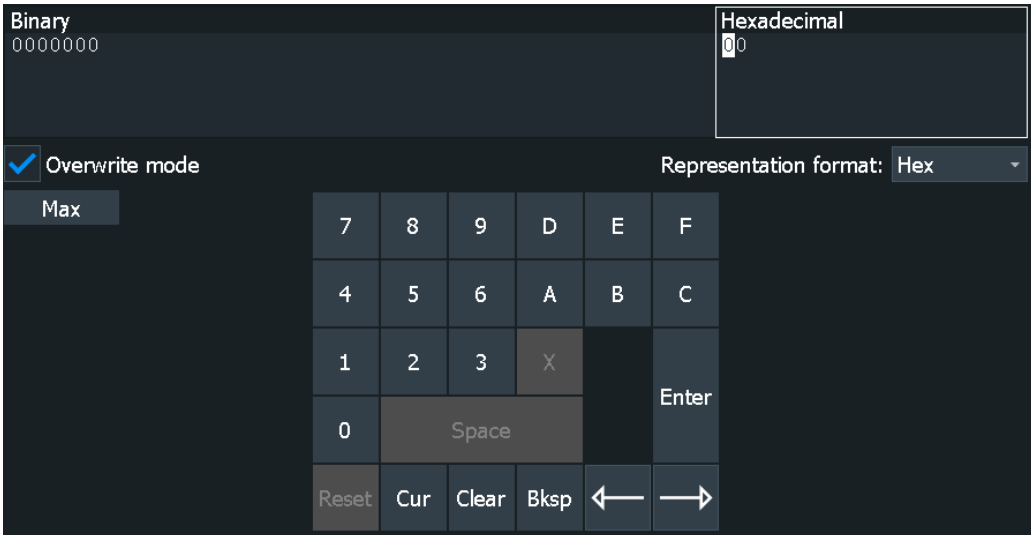
"Save"	Saves the selected results to the indicated file.
"Save As..."	Opens the file selection dialog box and saves the selected results to the selected file. See also Chapter 12.5, "File selection dialog", on page 478
File format	Selects the file format.

- ".csv": the values are saved in a file.
  You can select the value delimiter and the list separator symbol in the "CSV Export" dialog, see Chapter 4.8.2, "CSV export", on page 112.
  Tip for using MS Excel: It is recommended that you use the semicolon as csv decimal symbol. When you open the file with MS Excel, use "File > Open" and follow the wizard to set the separators correctly, or set the separator settings with "Tools > Options > International".
- ".html": the results are saved as web page for display in a browser.
- ".xml": the results are saved in an xml compatible file format.
- ".py": the values are saved in a python compatible file format.

Remote command:
`BUS<m>:EXPResult:SAVE` on page 1672

13.1.7 Bit pattern editor

If you want to enter a specified address or data pattern, the bit pattern editor helps you to enter the pattern in various formats - decimal, hexadecimal, octal, binary and ASCII.



The editor displays the pattern in two columns. The left column always shows binary data. For the right column, you can select the format, the default depends on the data specifics. You can edit data in the left or right column. The keypad adapts itself to the column format, only keys appropriate to the format are enabled.

The data is grouped and converted in bit groups. The size of a bit group depends on the address or data specifics and is set by the instrument. Groups are automatically separated by blanks. The maximum size of a bit group is 64 bit, the most common group size is 1 byte.

"Overwrite mode": If disabled, the data behind the new digit is shifted to the right. Bit groups are rearranged automatically.

Format-specific information:

- Unsigned: Decimal data format without sign. It is available for I²C, SPI, UART, CAN, LIN and FlexRay protocols. If you enter a decimal number that is too large for the defined bit group, the number is truncated and a message appears. X (do not care) in the decimal column sets all binary digits of the bit group to X.
- Signed: Signed decimal format, available for audio protocols. The first bit represents the sign. You can use the 2's complement or 1's complement format.
- Binary: 0, 1 and X (do not care) is allowed.
- Hex: most common format in the right column.
- Octal: Each digit represents 3 bit.
- ASCII: In the ASCII column, "X" is the character X. The binary X (do not care) is not allowed. If an X is included in the binary value in the left column, the ASCII column displays "§" to indicate that the value is not defined.

Where applicable, frequently used values are provided in a "Predefined values" list below the pattern table, for example, reserved end words of data packets in the UART protocol.

13.2 I²C (option R&S RTO-K1)

The Inter-Integrated Circuit is a simple, low-bandwidth, low-speed protocol used for communication between on-board devices.

13.2.1 The I²C protocol

This chapter provides an overview of protocol characteristics, data format, address types and trigger possibilities. For detailed information, read the "I2C-bus specification and user manual" available on the NXP manuals webpage at http://www.nxp.com/.

I²C characteristics

Main characteristics of I²C are:

- Two-wire design: serial clock (SCL) and serial data (SDA) lines
- Master-slave communication: the master generates the clock and addresses the slaves. Slaves receive the address and the clock. Both master and slaves can transmit and receive data.
- Addressing scheme: each slave device is addressable by a unique address. Multiple slave devices can be linked together and can be addressed by the same master.
- Read/write bit: specifies if the master reads (=1) or writes (=0) the data.
- Acknowledge: takes place after every byte. The receiver of the address or data sends the acknowledge bit to the transmitter.

The R&S RTO supports all operating speed modes: high-speed, fast mode plus, fast mode, and standard mode.

Data transfer

The format of a simple I²C message (frame) with 7-bit addressing consists of the following parts:

- Start condition: a falling slope on SDA while SCL is high
- 7-bit address of the slave device that either is written to or read from
- R/W bit: specifies if the data is written to or read from the slave
- ACKnowledge bits: is issued by the receiver of the previous byte if the transfer was successful
Exception: At read access, the master terminates the data transmission with a NACK bit after the last byte.
- Data: several data bytes with an ACK bit after every byte
- Stop condition: a rising slope on SDA while SCL is high

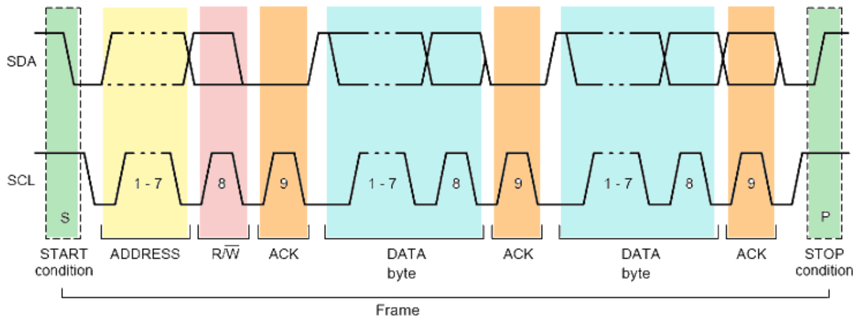


Figure 13-1: I2C writes access with 7-bit address

Address types: 7-bit and 10-bit

Slave addresses can be 7 bits or 10 bits long. A 7-bit address requires 1 byte, 7 bits for the address followed by the R/W bit.

A 10-bit address for write access requires 2 bytes: the first byte starts with the reserved sequence 11110, followed by the two MSB of the address and the write bit. The second byte contains the remaining 8 LSB of the address. The slave acknowledges each address byte.

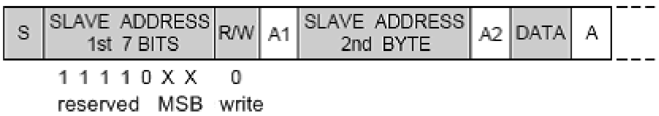


Figure 13-2: 10-bit address, write access

A 10-bit address for read access requires 3 bytes. The first 2 bytes are identical to the write access address. The third byte repeats the address bits of the first byte and sets the read bit.

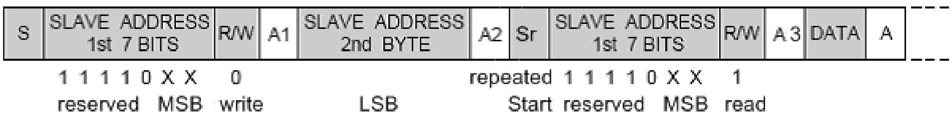


Figure 13-3: 10-bit address, read access

Trigger

The R&S RTO can trigger on various parts of I²C messages. The data and clock lines must be connected to the input channels, triggering on math and reference waveforms is not possible.

You can trigger on:

- Start or stop condition
- Repeated start condition
- Transfer direction (read or write)
- Bytes with missing acknowledge bit
- Specific slave address or address range

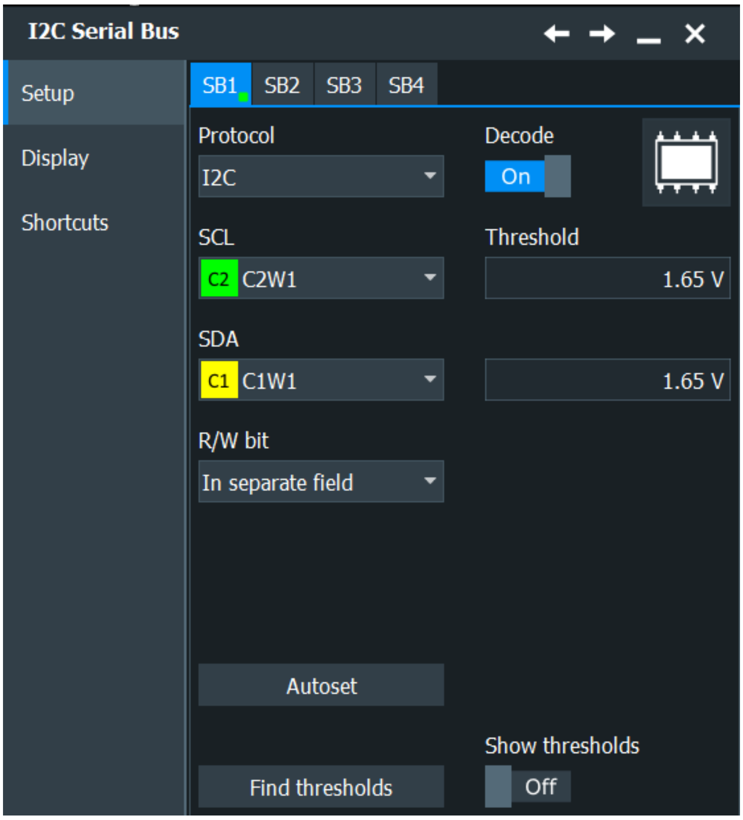
- Specific data pattern in the message

13.2.2 I²C configuration

13.2.2.1 I²C configuration settings

Access: [Protocol] > "Setup" tab > "Protocol" = *I2C*

Make sure that the tab of the correct serial bus is selected.



See also: Chapter 13.1.1, "Setup - general settings", on page 481

SDA, SCL

Set the waveforms of the data line (SDA) and clock line (SCL).

Usually, the source is one of the analog channels. Reference and math waveforms are available as source if the trigger source is one of the analog channels but not the serial bus.

Do not combine a reference waveform with channel or math waveform because the time correlation of these waveforms might differ.

Alternatively, digital channels can be used if MSO option R&S RTO-B1 is installed. Digital and analog channels cannot be used at the same time.

For triggering on a serial bus, analog or digital channel sources are required.

Remote command:
BUS<m>:I2C:SDA:SOURce on page 1675
BUS<m>:I2C:SCL:SOURce on page 1675

R/W bit
Defines if the R/W bit is considered separately or as part of the address. The setting affects the Adress Type of the trigger conditions.

Remote command:
BUS<m>:I2C:RWBit on page 1677

Threshold
Sets the threshold value for digitization of signals for each line. If the signal value on the line is higher than the threshold, the signal state is high (1 or true for the Boolean logic). Otherwise, the signal state is considered low (0 or false) if the signal value is below the threshold.

There are several ways to set the threshold:
- "Threshold"
  Enter the value directly in the field.
- "Autoset"
  Starts software algorithms for determining the signal threshold levels and bitrate.
- "Find thresholds"
  Executes the measurement of reference levels and sets the thresholds to the middle reference level of the measured amplitude.

Note: If the sources are digital channels, the same threshold values are used for the parallel and the serial buses. You can set the thresholds either in the parallel bus configuration or in the serial bus configuration.

Remote command:
BUS<m>:I2C:SCL:THReshold on page 1676
BUS<m>:I2C:SDA:THReshold on page 1676
BUS<m>:I2C:TECHnology on page 1676
BUS<m>:SETReflevels on page 1670
BUS<m>:FAUToset on page 1670

Show thresholds
If enabled, the threshold levels are displayed in the diagram.

Remote command:
BUS<m>:THReshold on page 1670

13.2.2.2 Display settings

Access: [Protocol] > "Display" tab.

For common display settings, see Chapter 13.1.3, "Display", on page 483.

I2C Specific Label Settings

Label lists are protocol-specific. Label lists for I²C are available in CSV and PTT format.

An I²C label file contains three values for each address:

- Address type, 7-bit or 10-bit long
- Address value
- Symbolic label: name of the address, specifying its function in the bus network.

Example: I²C PTT file

```
# ----------------------------------------------------------------------------
@FILE_VERSION  = 1.00
@PROTOCOL_NAME = i2c
# ---------------------------------------------------------------------------
# Labels for I2C protocol
#   Column order: Identifier type, Identifier value, Label
# ---------------------------------------------------------------------------
7,0x1E,Voltage
7,38h,Pressure
7,2Ah,Temperature
7,16h,Speed
7,118,Acceleration
7,07h,HighSpeed_Master_0x3
7,51h,EEPROM
10,3A2h,DeviceSetup
10,1A3h,GatewayStatus
10,06Eh,LeftSensor
# ----------------------------------------------------------------------------
```

13.2.2.3 Configuring I²C protocol

The configuration of the I²C is simple - assign the two lines to input channels, and set the thresholds.

For details on configuration settings, see Chapter 13.2.2, "I²C configuration", on page 492.

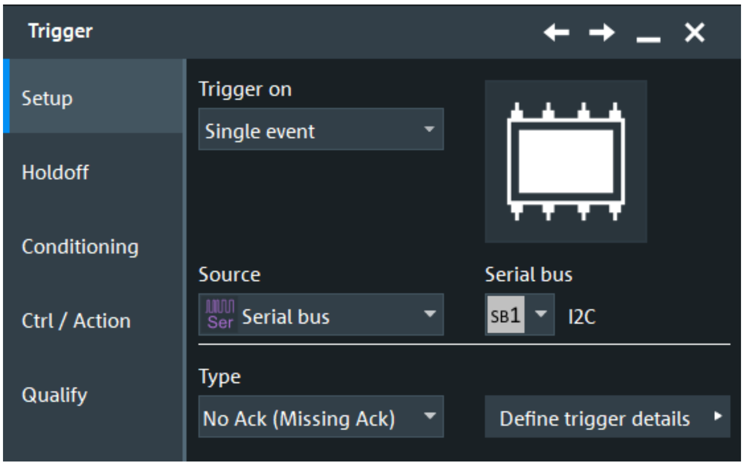
1. Press the [Protocol] key on the front panel.
2. Select the tab of the bus you want to set up, for example "SB1".
3. Select the "Setup" tab.
4. Tap the "Protocol" button and select the protocol: "I2C".
5. Tap the "SDA" button, and select the waveform of the data line.
6. Tap the "SCL" button, and select the waveform of the clock line.
7. Set the logical thresholds.

8. Enable "Decode".

13.2.3 I²C trigger

13.2.3.1 I²C trigger settings

Access: [Protocol] > "Shortcuts" tab > "Setup trigger"



Make sure that:

- In the [Protocol] > "Setup" tab:
  - The correct "Protocol" is selected.
  - The data sources of the serial bus are channel signals.
- In the "Trigger" > "Setup" tab:
  - The "Trigger on" = "Single event"
  - The trigger "Source" = "Serial bus".
  - The correct "Serial bus" is selected.

Serial bus
Selects the serial bus to be triggered on. Make sure to select the correct bus before you enter the settings.

To trigger on a serial bus, the signals sources must be channel signals. If the data or clock source is a math or reference waveform, you cannot trigger on that bus.

Remote command:
`TRIGger<m>:SOURce:SBSelect` on page 1674

Trigger type
Selects the trigger type for I²C analysis.

Some trigger types have additional settings that can be defined. In this case, the "Define trigger details" button appears next to the "Type" function. Open it for a detailed definition of the trigger conditions.

Remote command:
`TRIGger<m>:I2C:MODE` on page 1678

Start ← Trigger type
Sets the trigger to the start of the message. The start condition is a falling edge on SDA while SCL is high. The trigger instant is the falling edge of the SDA line.

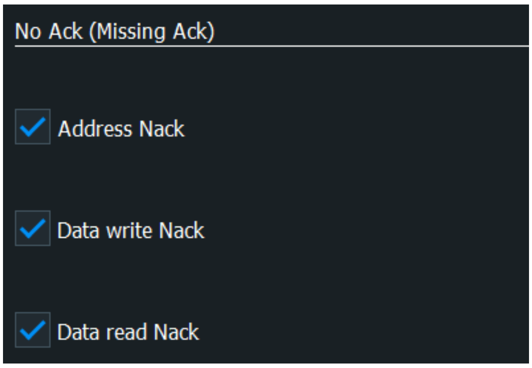
You can change the SDA and SCL lines here if necessary.

Repeated start ← Trigger type
Sets the trigger to a repeated start - when the start condition occurs without previous stop condition. Repeated start conditions occur when a primary exchanges multiple messages with a secondary without releasing the bus.

Stop ← Trigger type
Sets the trigger to the end of the message. The stop condition is a rising slope on SDA while SCL is high.

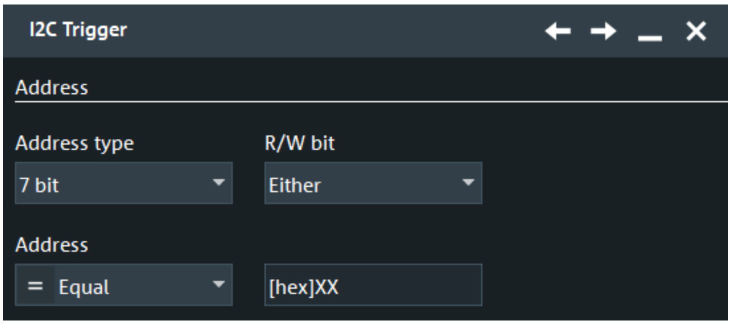
No Ack (Missing Ack) ← Trigger type
Missing acknowledge bit: the instrument triggers if the data line remains HIGH during the clock pulse following a transmitted byte.

You can also localize specific missing acknowledge bits by setting the No Ack conditions.



Address ← Trigger type
Sets the trigger to one specific address condition or a combination of address conditions. The trigger time is the falling clock edge of the acknowledge bit after the address. You can specify:

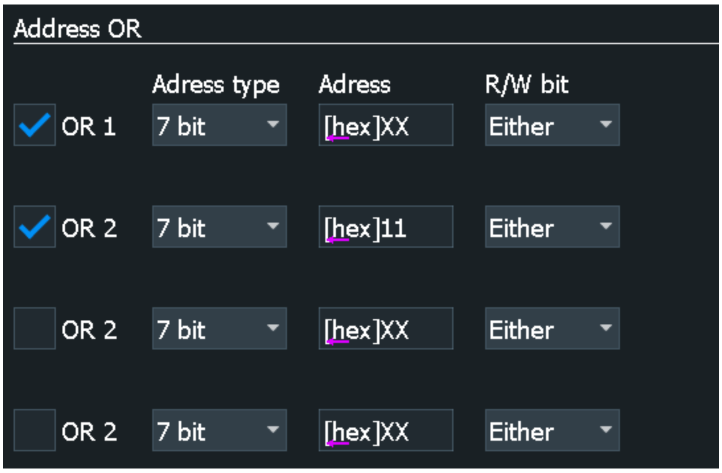
- Adress Type
- "Address" on page 498
- R/W bit



Address OR ← Trigger type
Triggers on one to four address conditions. Each enabled condition requires an exact address. You can specify:

For each condition to be used, select "Monitor"

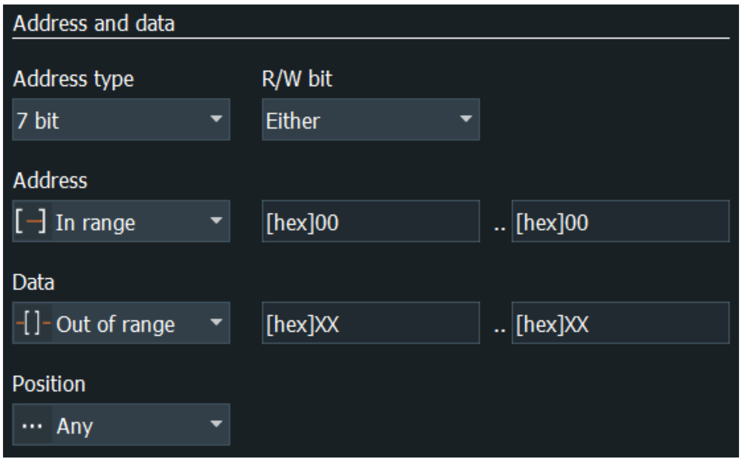
- "OR x" on page 499
- Adress Type
- "Address" on page 498
- R/W bit



Address and data ← Trigger type
Sets the trigger to a combination of address and data condition. You can specify:

- Adress Type
- R/W bit
- "Address" on page 498
- "Position" on page 499
- "Data" on page 499

The address conditions are the same as for the "Address" trigger type and "Data" on page 499.



No Ack conditions

Selects which missing acknowledge bits is detected if the trigger type is set to "Missing Ack".

"Address Nack"
: No secondary recognizes the address.

"Data write Nack"
: The addressed secondary does not accept the data.

"Data read Nack"
: Marks the end of the read process when the primary reads data from the secondary. This Nack is sent according to the protocol definition, it is not an error.

Remote command:
`TRIGger<m>:I2C:ADNack` on page 1679
`TRIGger<m>:I2C:DWNack` on page 1679
`TRIGger<m>:I2C:DRNack` on page 1679

Adress Type

Sets the address length to be triggered on: 7 bit, 7+1 bit, or 10 bit. Available settings depend on the R/W bit setting of the bus configuration.

For "7 bit" and "10 bit", enter the address bits in the Address field, and use the "R/W bit" on page 499 field to select the transfer direction.

For "7+1 bit", enter the seven address bits and also the R/W bit in the "Address" field.

If the trigger type is "Address + data", you can set the address type "Any" to trigger on data only, regardless of the address.

Remote command:
`TRIGger<m>:I2C:AMODe` on page 1679
`TRIGger<m>:I2C:ADOR<n>:ADRType` on page 1681

Address

The trigger address setup consists of a comparison condition and one or two address patterns.

Defines the bit pattern of the secondary device address. The length of the entry is adjusted to the selected address type.

In binary format, use the following characters: 1; 0; or X (do not care). To enter the pattern in other formats use the bit pattern editor, see Chapter 13.1.7, "Bit pattern editor", on page 488.

"Condition"	Sets the comparison condition to a specific value or a range.
"Min"	Specifies the value or sets the start value of a range.
"Max"	Sets the maximum value of a range for "Condition" = "In range"/"Out of range".

Remote command:
`TRIGger<m>:I2C:ADDRess` on page 1680
`TRIGger<m>:I2C:ADDTo` on page 1680
`TRIGger<m>:I2C:ADOR<n>[:VALue]` on page 1681

R/W bit
Toggles the trigger condition between read and write access of the primary. Select "Either" if the transfer direction is not relevant for the trigger condition.

Remote command:
`TRIGger<m>:I2C:ACCess` on page 1678
`TRIGger<m>:I2C:ADOR<n>:RWBit` on page 1681

OR x
Enables each selected condition, for "Trigger Type" = "Address OR".

Remote command:
`TRIGger<m>:I2C:ADOR<n>:ENABle` on page 1680

Position
Sets the number of data bytes to be skipped after the address. You can define an exact position, or a position range. The index 0 is associated with the first data byte.

"Condition"	Sets the comparison condition to a specific value or a range.
"Index min"	Specifies the value or sets the start value of a range.
" Index max"	Sets the maximum value of a range for "Condition" = "In range"/"Out of range".

Remote command:
`TRIGger<m>:I2C:DPOPerator` on page 1682
`TRIGger<m>:I2C:DPOSition` on page 1682
`TRIGger<m>:I2C:DPTO` on page 1682

Data
Specifies the trigger conditions for the data bit pattern.

In binary format, use the following characters: 1; 0; or X (do not care). To enter the pattern in other formats use the bit pattern editor, see Chapter 13.1.7, "Bit pattern editor", on page 488.

"Condition"	Sets the comparison condition to a specific value or a range.

"Value min "	Specifies the value or sets the start value of a range. Enter the bytes in msb first bit order. The maximum pattern length is 64 bit. Waveform data is compared with the pattern byte-by-byte.
"Value max "	Sets the maximum value of a range for "Condition" = "In range"/"Out of range".

Remote command:
TRIGger<m>:I2C:DCONdition on page 1682
TRIGger<m>:I2C:DMIN on page 1683
TRIGger<m>:I2C:DMAX on page 1683

13.2.3.2 Triggering on I²C signals

Prerequisites: An I²C bus is configured, see Chapter 13.2.2.3, "Configuring I²C protocol", on page 494.

1. Press the [Protocol] key and select the "Shortcuts" tab.
2. Press "Setup trigger".
3. Tap the "Source" button and select the "Serial bus" trigger source.
4. Select the serial bus that is set to I²C.
5. Select the "Trigger type".
6. For more complex trigger types, enter the address and/or data conditions: address, acknowledge bits, R/W bit, and data pattern.
   For details, see Chapter 13.2.3, "I²C trigger", on page 495

13.2.4 I²C decode results

When the configuration of the serial bus is complete, the signal can be decoded:

1. In the "Serial Bus" dialog > "Setup" tab, enable "Decode".
2. In the "Serial Bus" dialog > "Display" tab, select additional result display settings: "Show table" and "Show binary".
3. If required, enable "Zoom coupling"

For a description of the display settings, see also Chapter 13.1.3, "Display", on page 483.

The instrument captures and decodes the signal according to the standard definition and the configuration settings.

The color-coding of the various protocol sections and errors simplifies the interpretation of the visual display. The decode information condenses or expands, depending on the horizontal scale. Various data formats are available to show the result values.